EUROVENT 1/12

Sources of Error in Aerodynamic System Resistance and Acoustic Calculation

EUROVENT 1/12

First Edition - 2011

This document has been prepared by Eurovent WG 1 “Fans” based on a draft written by its Chairman, Bill CORY. Important contributions have been provided by Alain GUEDEL (CETIAT), Dario BRIVIO (Nicotra), Hans WITT (Witt & Sohn), and Alain GODICHON (FlaktWoods). The text has been extensively discussed by all the other active members of the group. Special thanks go to Ron MULHOLLAND (Howden) for the high quality drawings.

Bill CORY

Alain GODICHON

Ron MULHOLLAND

Hans WITT

Alain Guedel

Dario BRIVIO

Published by EUROVENT
Diamant Building, Bd Reyers 80
1030 Brussels, Belgium
Supervised by Nina ION
nina.ion@eurrovent-association.eu

Contents

Chapter 1 Influence of the Boundary Layer 6
1.1 Synopsis 6
1.2 Introduction 6
1.3 Friction factors and Reynolds Numbers 8
1.4 Moody Charts 9
1.5 Laminar Flows 11
1.6 Transitional Flows 12
Chapter 2: Bends.. 13
2.1 Introduction 13
2.2 Losses in the transitional regime 13
2.3 Effect of velocity profile and turbulence level downstream of a singularity............14
Chapter 3: The Influence of Temperatures Differing from Standard 14
Chapter 4: Height Effects 16
Chapter 5: Interaction between Resistances 16
5.1 Introduction 16
5.2 Energy loss coefficients of two 90° elbows separated by a straight duct 17
5.3 Measured data 19
5.4 Comparison between the measured and predicted data 19
5.5 Conclusions 20
Chapter 6: Upstream and Downstream Flow Conditions 21
Chapter 7: Swirling Flow and the Flow Profile 24
Chapter 8: Differences between planned and built installations 26
Chapter 9: Fan Noise Installation Effects 30
9.1 Introduction 30
9.2 Examples of fan noise system effect 30
9.2.1 Fan coupled to a heat exchanger 30
9.2.2 Abrupt bend at the inlet of a tubeaxial fan 30
9.3 Prediction of fan noise system effect 32
9.3.1 Experimental set-up and test result 32
9.3.2 Prediction of the acoustic system effect 33
9.4 Conclusions 35
9.5 References 35
Chapter 10: Installation Effects of a Fan coupled to a Heat Exchanger 36
10.1 Introduction 36
10.2 Origin of the fan installation effects 36
10.2.1 Aerodynamic system effects 36
10.2.2 Acoustic system effects 37
10.3 Axial fan in front of a finned tube heat exchanger 37
10.4 Influence of the distance between a cooling fan and a radiator in agricultural machinery application..39
10.5 Quantification of system effects of engine cooling fan on mobile machinery.....41
10.6 Conclusions 42
10.7 References 42
Annex A: System resistance – Interaction with combustion.................................. 44
A.1 Combustion 44
A.2 Operating advantages 44
A.3 Determining the correct fan duty..45

Foreword

Even a careful pre-calculation of the expected pressure losses in a system may lead to results that differ considerably from the actual values found in the system when built. This may have negative effects on the power consumption and the noise level of the installation. Under or over evaluation of losses will lead to airflows differing from the design optimum with negative cost implications and adverse consequences for the processes concerned.

In the worst cases, under-evaluation of losses may cause the supply fan to go into stall, which can lead to a premature destruction of the fan (see Eurovent Document 1/11).

Some of the most common sources of error are detailed below. Where possible, information is included to enable a more exact pre-calculation of losses.

a. Influence of the boundary layer
b. Temperature influences
c. Upstream flow disturbances
d. Interactions between different resistances
e. Interactions between different elements such as combustion engines and fans
f. Differences between the system as planned and as built
g. Swirl and distorted flow profiles

Chapter 1 Influence of the Boundary Layer

1.1 Synopsis

For many years we have lived with the statement that the system resistance is proportional to the flow rate squared i.e. that it is a quadratic relationship. Whilst this is almost true for small changes in the flow, its shortcomings are beginning to be realized in an era of inverters and other methods of wide ranging speed control. System air velocities are often in the transitional zone, such that Reynolds Number effects have to be considered. The wider the range of flows, the bigger the observed discrepancies. This paper gives the theoretical reasons for the breakdown of the 'square law'. It also notes that there is no intrinsic loss coefficient for a given duct fitting.

1.2 Introduction

In the design of a ductwork system it is the practice to add the resistance of all the elements in the duct run having the largest resistance (i.e. the index leg) together, to determine the total (or static) pressure loss. Allowing for system effects, the fan must develop this pressure at the design flow rate. The system and fan will then be in harmony.

The resistance of duct fittings and straight ducting are invariably determined from the Guides produced by bodies such as ASHRAE or CIBSE. However, readers of this document are referred to the Eurovent publications 2/9 Experimental determination of mechanical energy loss coefficients of Air handling components and 2/10 Catalogue of energy loss coefficients of air handling components which detail the results obtained from a collaborative programme within Europe, conducted by four independent laboratories. Most organizations have a similar approach and treat the pressure losses as a function of the local velocity pressure. This function is usually regarded as a constant and thus the loss becomes:

$$p_{Lf} = \varsigma_F \times \frac{1}{2}\rho v^2 \qquad \textit{Equation 1}$$

where:

p_{Lf} = pressure loss (Pa)
ζ_F = resistance coefficient
ρ = local air density (kg/m^3) (usually taken as standard 1.2)
v = 'average' velocity (m/s) = $\dfrac{\text{flowrate } q_v}{\text{cross sectional area } A}$

Whilst this may be reasonably true in the normal working range, it is important to know that ζ_F has a Reynolds Number dependence and that at low Reynolds Numbers ζ_F can increase enormously, whilst in fully turbulent flow, if ever attained, the value could be less.

There is no intrinsic value of the energy loss coefficient for an air-handling component. It is important to realise that for each and every upstream flow condition a different value may be found. In most guides and textbooks a fully developed symmetrical velocity profile without swirl is assumed at the entry to the component and that this is recovered downstream. This requires the use of a long straight duct both sides of the component. Such a condition is frequently unobtainable in practice. Even different lengths of these ducts and different entry conditions can produce significant variations in the flow pattern and hence the real energy loss resistance coefficient. It will of course be noted that at very low velocities, when the flow is laminar, the air velocity profile has a definite peak with a maximum velocity as much as twice the average. Hence the coefficient will be much higher, albeit acting on a very low velocity pressure.

There are very few text books which ever admit this variation. Certainly past editions of the CIBSE and ASHRAE guides failed to note any such problems. However, following the publication of a paper by Peter Koch, the dependence on both size and Reynolds Number is now accepted by CIBSE and SMACNA.

Thus the loss in a fitting is more correctly given by the expression

$$p_{Lf} \;=\; \varsigma_F \;\times\; f_1(\mathrm{Re}) \;\times\; f_2(d) \;\times\; \tfrac{1}{2}\rho v^2 \qquad \textit{Equation 2}$$

where f_1 and f_2 are correction factors depending on Reynolds Number and diameter respectively.

It might be thought that the topic is somewhat esoteric, but it is suggested that with the increasing use of inverters and other variable flow devices, **it is important to know that at high turndown ratios, the system resistance curve diverges ever more from the often quoted that "p_L" is proportional to Q^2.** Thus power absorbed is **not** proportional to fan speed n^3, even if there were no bearing, transmission or control losses. (see ISO 5801: 2007 Annex E)

In like manner, the loss in straight ducting is usually quoted as:

$$p_{Ls} = \frac{fL}{m} \cdot \frac{1}{2}\rho v^2 \qquad \textit{Equation 3}$$

and $\frac{fL}{m}$ is taken to be a constant ζ_s

where:

L = length of straight duct (m)
m = mean hydraulic depth (m) = cross sectional area ÷ perimeter

$= \frac{d}{4}$ for circular cross-sections

f = friction factor

Again, as L and m are constants and f is assumed to be constant, the loss is taken to be

$$p_{Ls} = \zeta_s \cdot \frac{1}{2}\rho v^2 \qquad \textit{Equation 4}$$

And thus another problem is created, for f is **not** a constant but rather a function of size, absolute roughness and Reynolds Number.

1.3 Friction factors and Reynolds Numbers

A number of prominent researchers have been associated with the development of charts to show a relationship between friction factor and Reynolds Number. Amongst those most widely known are Prandtl, von Kármán, Nikuradse, Poisseuille, Colebrook and White. Fig. 1 is taken from German literature.

More recently Moody was able to take the Colebrook-White and Poisseuille data to produce the well-known Moody chart of 1944. This has subsequently been used by ASHRAE, CIBSE, and AMCA International in their publications.

The derivation of pressure loss factors should be appreciated. Many of the sources give conflicting values. This is not surprising considering that much of the experimental work dates back more than 100 years

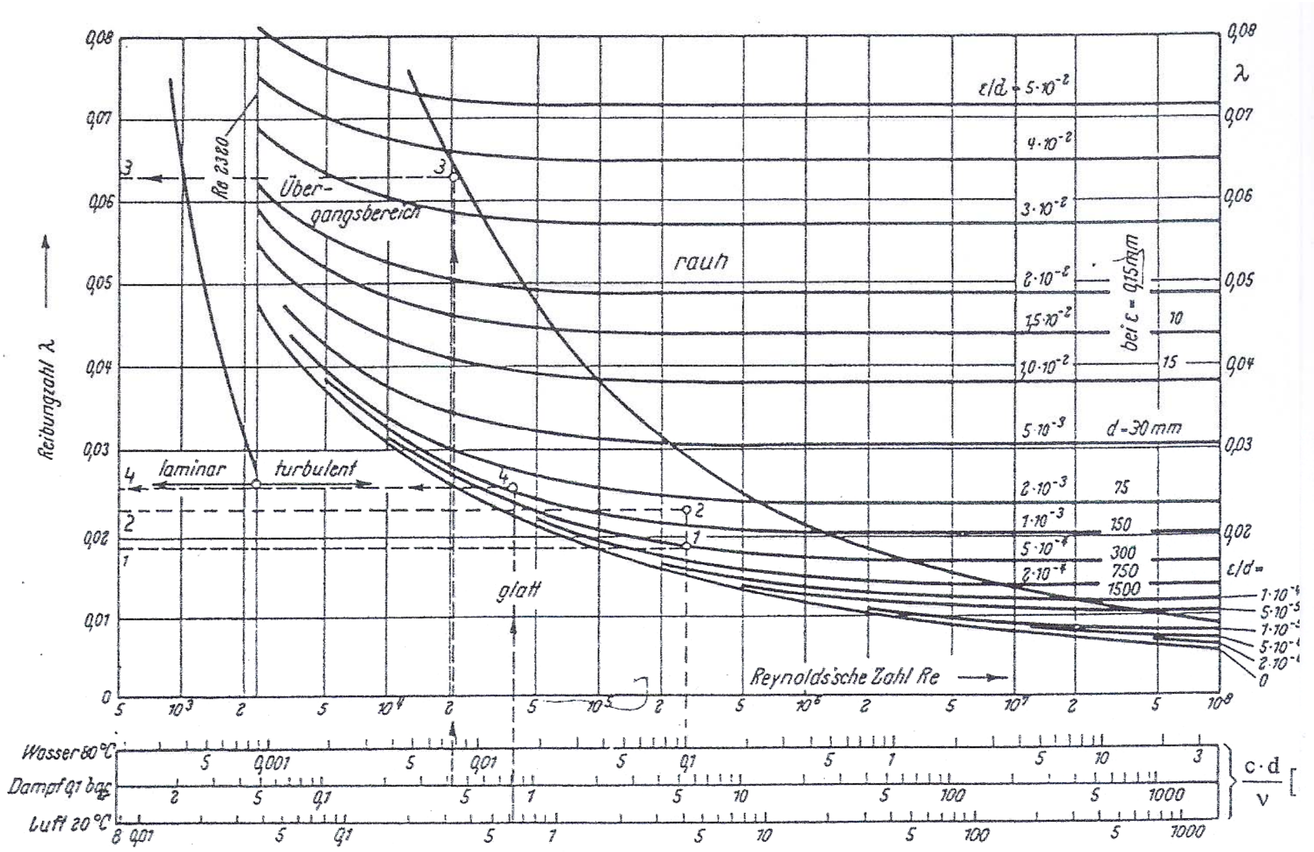


Fig. 1

1.4 Moody Charts

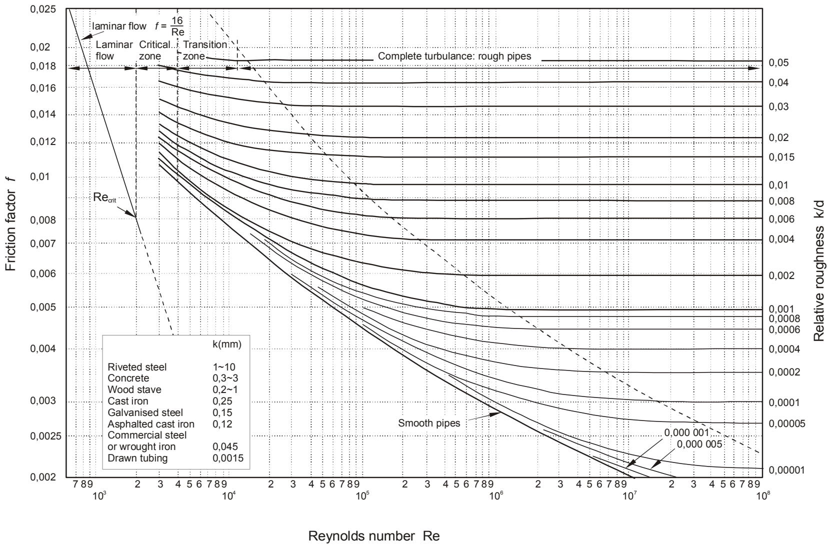


Fig. 1a: Friction factor versus Reynolds number – Moody chart (European)

The Moody chart Fig. 1a shows that in the transitional and lower zones f ≠ constant, and that again, as flow is diminished and enters the critical zone there are significant increases in f, then a sudden drop, before climbing again in the laminar zone. The friction factor f is only constant to the right of the dotted line.

Here, we have to interject with a warning. Not only are Europe and America divided by units, but also by definitions. In the U.S.A. most textbooks define the friction loss of straight ducting in terms of d – its diameter. We use the mean hydraulic depth m. Thus as m = d/4, the values of f are multiplied by 4 (see AMCA 200-95 p9) and the critical values of Reynolds N^{o} may also be changed.

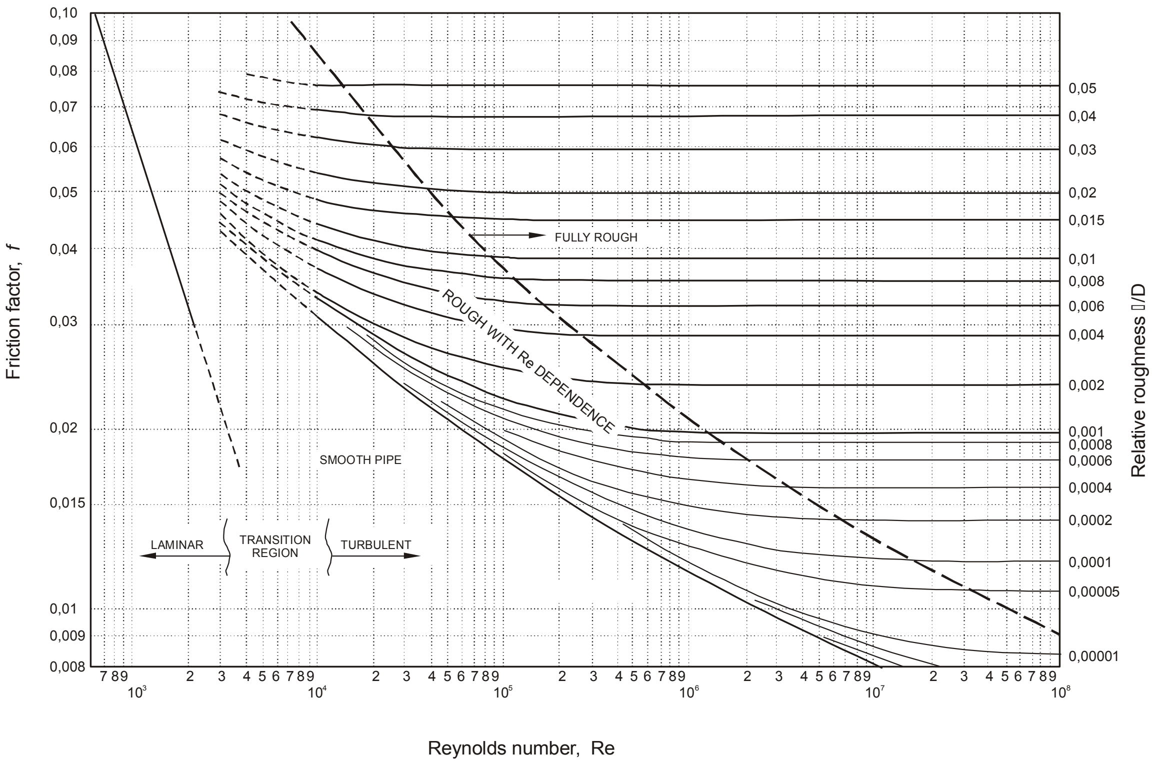


Fig. 1b: Friction factor versus Reynolds Number - Moody chart (American)

It is interesting and instructional to note that for the range of duct sizes and velocities encountered in normal ventilation systems the flow is always in the transitional zone. Only in the largest sizes and at the highest velocities does the flow even approach full turbulence. This is shown in Table 1 below.

Diameter d m	**Average Velocity v m/s**	**Reynolds No.** $Re = \frac{\rho v d}{\mu}$	**Relative Roughness** $\frac{k}{d}$	**Friction Factor f**	**Flow quality**
0.1	2.5	16492	0.0015	0.0076	Transitional
	5	32985		0.0067	
	10	65970		0.0063	
	15	98955		0.0059	
	20	131940		0.0057	
0.25	2.5	41231	0.0006	0.006	Transitional
	5	82463		0.0055	
	10	164926		0.005	
	15	247388		0.0048	
	20	329851		0.0047	
0.315	5	103903	0.00048	0.0051	Transitional
	10	207806		0.0047	
	15	311710		0.0046	
	20	415613		0.0045	
	25	519516		0.0044	
0.63	5	207806	0.00024	0.0043	Transitional
	10	415613		0.0042	
	15	623419		0.0039	
	20	831226		0.0038	
	25	1039032		0.0036	
1	5	329851	0.00015	0.0039	Transitional
	10	659703		0.0037	
	15	989555		0.0036	
	20	1319406		0.0035	
	25	1649258		0.0034	
2	10	1319406	0.000075	0.0033	Transitional
	15	1979109		0.0032	
	20	2638812		0.0031	
	25	3298516		0.003	
	30	3958218		0.00295	
2.5	15	2473887	0.00006	0.00295	Transitional
	20	3298516		0.0029	
	25	4123144		0.00285	
	30	4947773		0.0028	
	40	6597031		0.0028	

Table 1: Friction factors versus duct size and velocity (European conventions)

Note 1: Values apply to standard air

Note 2: All values are in the transitional range

Assuming an absolute roughness applicable to g.s.s. (galvanized sheet steel), it can be seen that in all these cases the flow is transitional. In this connection 'absolute roughness' is defined as the height of the roughness divided by the duct diameter and 'transitional' is defined as any flow where there is not complete turbulence, i.e., f is not constant over a range of Reynolds Nos. The relative roughness and friction factor therefore vary enormously as shown. Thus with decreasing flow, and thus velocity, the reducing velocity pressure is partially offset by the increase in f.

It should here be noted that the straight duct friction charts popular in many text books and guides invariably assume a constant value for 'f' and also a fully developed, swirl free velocity profile. They are therefore approximations only. The shaded area in fig 1c is likely to be nearer to the truth but only where the assumptions are also valid. As these conditions are seldom met in real installations an error of up to 50% may easily occur.

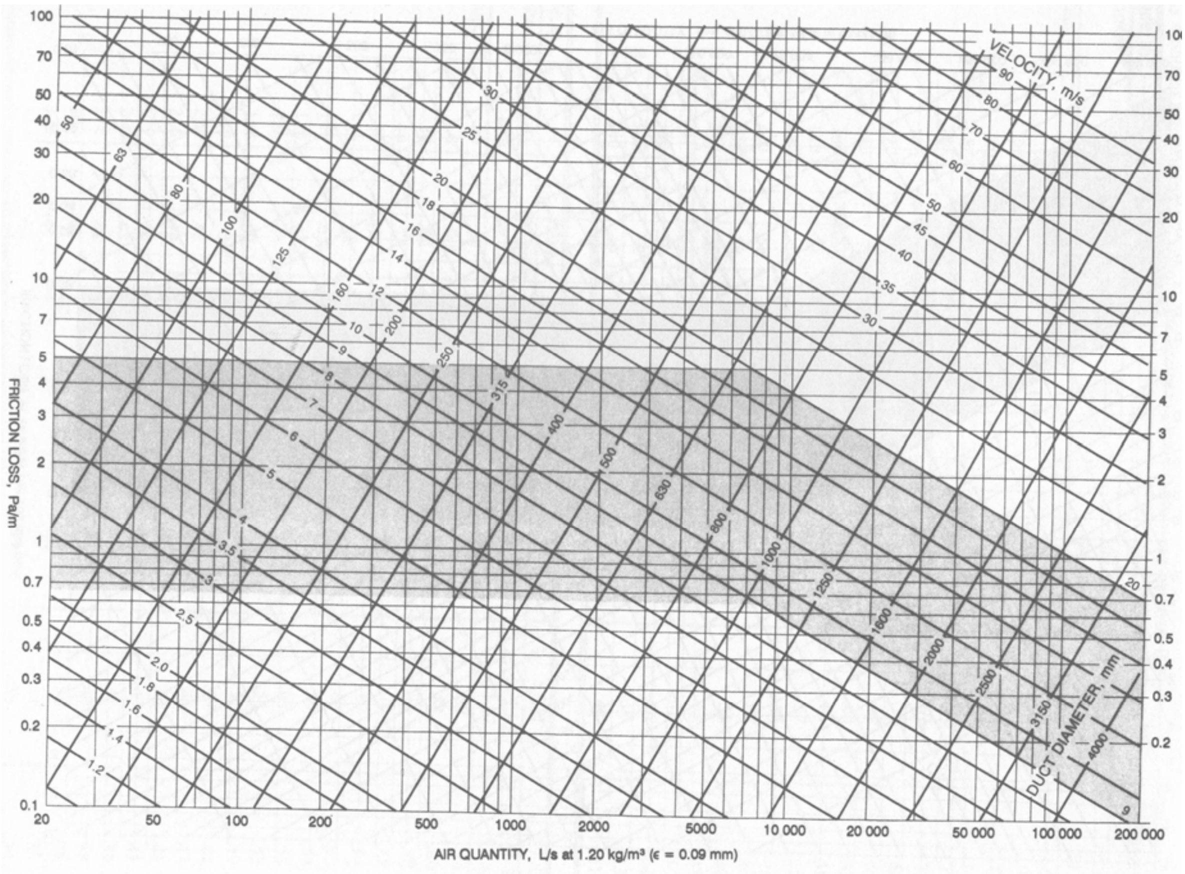


Fig. 1c Straight duct friction losses

1.5 Laminar Flows

At sufficiently low velocities and small duct sizes the Reynolds Number will be below 2000. The flow becomes laminar away from disturbances. Laminar flow may be defined as where all air velocity vectors are parallel to the duct walls and is essentially 'smooth' Only viscous forces are of any importance, making shear and hence energy dissipation directly proportional to velocity. In pressure loss terms we may say that Equation 1 is still valid. However the loss coefficient ζ_F for a particular component is then inversely proportional to the local Reynolds Number.

Therefore:

$$\varsigma_F = \frac{\text{Constant}}{\text{Reynolds Number}} \qquad \textit{Equation 5}$$

Within duct fittings abrupt changes in area and direction induce turbulence at Reynolds numbers well below 2000. Once turbulence is induced the fitting's loss coefficient is no longer inversely proportional to Reynolds number, nor is it necessarily only mildly dependent on Reynolds number, as it will usually become at Reynolds numbers above 10^4.

Flow patterns can also vary and affect loss coefficients in the purely laminar region. Up to Reynolds numbers of about 10, based upon the maximum velocity within a component, 'creeping' flow without separation is possible at abrupt changes in area. At slightly higher Reynolds numbers inertia forces become important causing laminar separation, followed by laminar re-attachment. A further increase in Reynolds number results in the separated flow becoming turbulent.

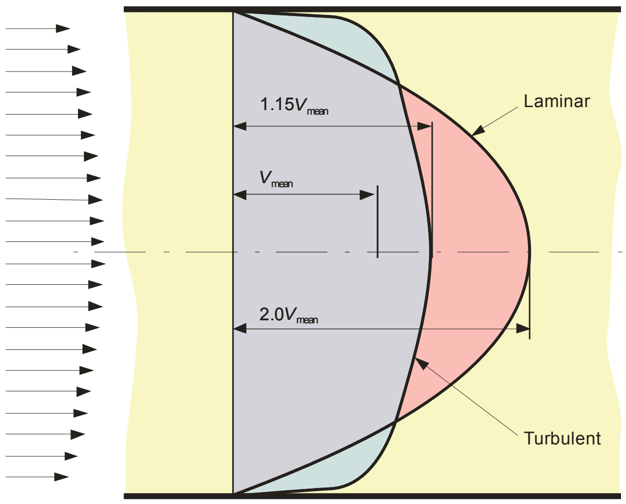


Fig. 2: Laminar and turbulent velocity profiles

Referring to Fig. 2, developed laminar flow is characterized by a kinetic energy coefficient and a peak to mean velocity ratio of 2, compared to developed turbulent profiles with a kinetic energy coefficient of just over unity and a peak to mean velocity ratio of about 1.2. If, due to turbulence within a component, an initially laminar flow leaves a component with a nearly uniform velocity, energy has to be taken from the mean flow in order to re-establish developed laminar flow. Following a smooth contraction the extra pressure loss over and above the friction loss calculated using developed flow friction coefficients, is 1.3 times the mean velocity pressure. Velocity reduction in laminar flow is accompanied by a greater energy dissipation than the theoretically recoverable velocity pressure, so diffusers have a reduced efficiency to the left of the dotted line in Fig. 1c and are completely useless in the laminar region.

1.6 Transitional Flows

The general shape of loss coefficient versus Reynolds number curves, for transitional flows, is shown in Fig. 3. The curves do not apply to smooth bends.

Chapter 2: Bends

2.1 Introduction

In most practical systems, the pressure losses in bends are the dominant factor and are usually far more important than the losses in straight ducting and other elements. Due to the relatively small surface area of a bend, the surface roughness may usually be neglected. Boundary layer problems i.e., Reynolds N^o. effects are of utmost importance. Any calculation of losses neglecting these can only be an approximation.

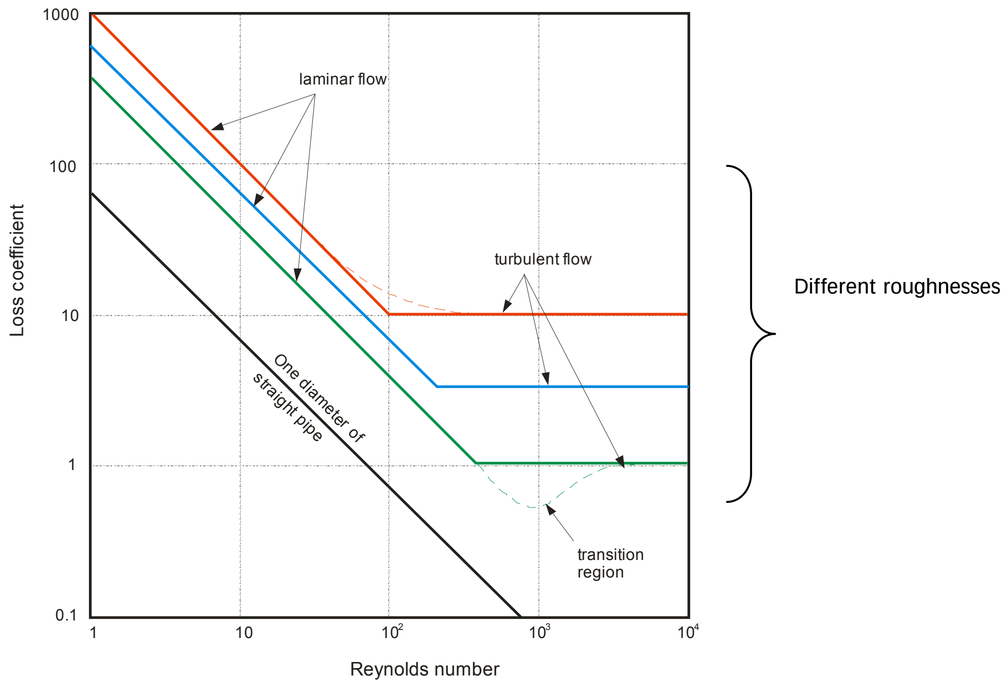


Fig. 3: Trends in loss coefficients in the transitional regime

2.2 Losses in the transitional regime

From Chapter 1, it is evident that the laminar to turbulent or transitional regime is the most complex. An example of how complex is again provided by the flow through 90° bends. Above Reynolds numbers of 100, applicable to virtually all air or gas systems, inertia forces become important. Centrifugal and static pressure forces acting on the highly peaked laminar velocity profile at inlet to a bend deflect the core region outwards. Secondary flows perpendicular to the main flow are set up in a similar manner to those with turbulent flow, but with laminar flow they are stronger and stable. The interaction of the secondary flows with the core region and the effects of flow stability on curvature, tend to delay the onset of turbulence to well above Reynolds numbers at which straight duct flow would become turbulent. At the same time as the onset of turbulence is being suppressed, the secondary flows grow in strength increasing viscous energy dissipation within the bend and in the outlet duct. Twin swirl can be set up and may be detached. These effects have been less well investigated and documented leading to larger insecurities in published results.

2.3 Effect of velocity profile and turbulence level downstream of a singularity

Because of the suppression described above, a turbulent inlet flow may become laminar, but highly distorted, and return to turbulent flow in the outlet duct. The complex phenomena that occur in the transition region are reflected in the shape of the bend loss coefficient curve in Fig. 4. Also shown in the figure is the approximate loss coefficient for an abrupt bend of centreline radius to diameter ratio of 0.7, which has a lower loss coefficient at Reynolds numbers below 1000 than larger radius ratio bends. It should be noted that the 'ledge' in the curves **may** be the normal 'constant' given in ASHRAE and previous CIBSE data.

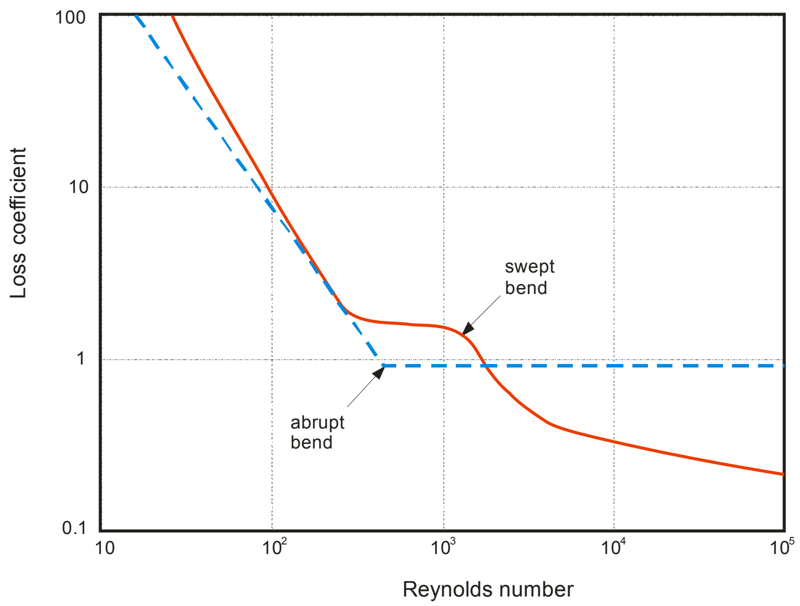


Fig. 4: Bend loss coefficients for transitional flows

Chapter 3: The Influence of Temperatures Differing from Standard

When dealing with air at temperatures other than standard, it is common practice to multiply the pressure loss in Pascals by the absolute temperature ratio. This is an approximation and merely allows for the effect of the change in air density. However a change in temperature also affects the absolute viscosity μ. Whilst this also falls with temperature, it is at a somewhat lower rate than the fall in air density. Thus the kinematic viscosity $\left(\frac{\mu}{\rho}\right)$ actually increases due to higher molecular velocities, which means that there is a reduction in Reynolds Number and a consequent increase in the friction factor f.

For small increments of temperature, the resulting error in f may frequently be neglected but, for higher temperatures, simply multiplying by the absolute temperature ratio i.e., the density ratio, may lead to very large errors, the magnitude of which cannot be assessed by a consideration of temperature alone.

As an example, the error in resistance value involved in neglecting the effect of change in f at 83°C is approximately 1.5% at 50 m/s, but 7% at 2 m/s. The error rises rapidly with temperature and at 540°C is approximately 10.5% at 50 m/s but 33% at 5 m/s.

The temperature correction chart, Fig. 5 enables an overriding correction to be applied to friction losses obtained from normal sheet steel ducting friction charts, after these have been corrected for the modified air density, as in common practice. It will be found that the two corrections oppose one another. Thus the omission of the correction given by Fig. 5 would lead to an optimistic (i.e. lower) estimate of friction loss. In addition, it should be noted that the actual velocity of air in the duct, at the elevated temperature should be used.

Example:

Actual velocity in the duct at temperature	=	20 m/s
Duct diameter	=	355 mm
Temperature of air	=	260°C
Friction loss at standard temperature	=	500Pa say
Correction factor from Fig 5 K_T	=	1.1

Friction loss at 260°C $= 500 \times \frac{273+20}{273+260} \times 1.1$ = 302 Pa

This example shows that by ignoring viscosity effects, the friction will have been underestimated by 10%. It should be noted that the chart cannot be accurately used in conjunction with Roughness Correction charts because the latter are based on particular friction factor ratios corresponding to Reynolds Numbers relating to standard air.

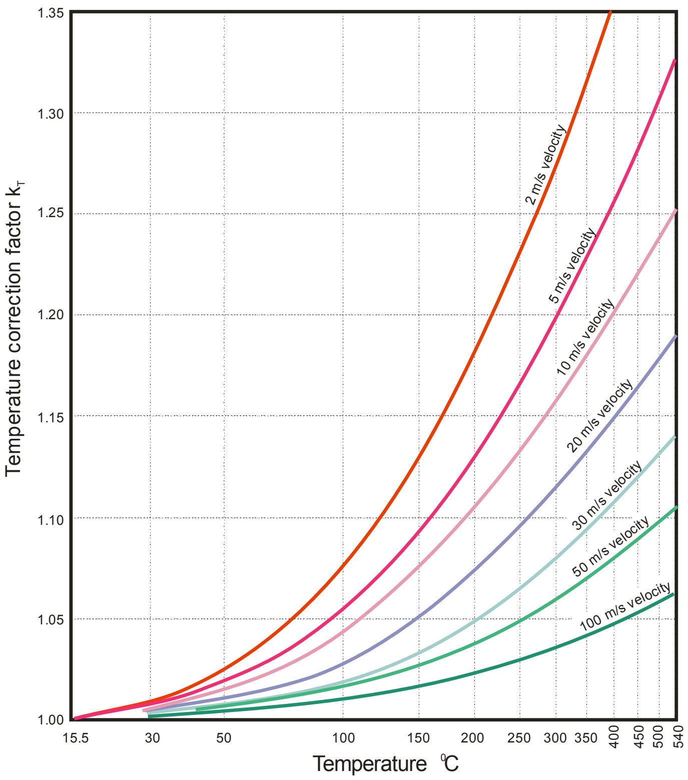


Fig. 5 Temperature correction chart for duct friction due to viscosity and Reynolds Number changes

Chapter 4: Height Effects

There is another effect which has to be considered when the air or gas within the fan and duct system differs from the surrounding ambient atmosphere. This from experience is rarely considered. Thus, where there is a difference in height above datum between the fan system inlet and outlet, a chimney stack effect will be created. This may be calculated from the formula:

$$p_{ce} = g\ h\left(\rho_{fs} - \rho_a\right) \qquad \textit{Equation 6}$$

Where:

p_{ce}	=	chimney stack effect	Pa
g	=	acceleration due to gravity	m/s^2
ρ_{fs}	=	density of gas in fan system	kg/m^3
ρ_a	=	density of ambient air	kg/m^3

Obviously, if the outlet is above the inlet to the system, the fan will be assisted and this amount should be deducted from the calculated system resistance. If, however, the inlet is at the higher point of the system, then this stack effect will oppose the fan and should be added to the normally calculated system resistance.

Again, it is not impossible that the air in the fan system is of lower temperature, and therefore higher density than that of the ambient air. In such cases the stack effect will be reversed. Thus in the case where the outlet is higher than the inlet, the system resistance will be increased by this figure (ignoring the sign). Where the inlet is higher than the outlet then the system resistance will be reduced (again ignoring sign).

In practice, air conditioning units are often placed at the top of the building. Thus for high rise buildings the error may be considerable especially with seasonal variations.

Chapter 5: Interaction between Resistances

5.1 Introduction

We have noted many times throughout this document that the total pressure loss for fan selection can be deduced by summating the losses of individual elements in the index leg of our system. This is easier said than done. As stated in Eurovent 2/9, there is no intrinsic value of energy loss coefficient for any component. For each upstream flow condition a different value will be found. The usual convention in any test work has been to use a long straight duct both upstream and downstream of the element such that a fully developed profile, free from swirl, is both presented to, and regained after the element. From these conventions, it will be immediately recognised that the assumption that the pressure loss in a component, assumed as $p_{LF} = \zeta_F$ x velocity pressure, can only be correct under very narrow conditions. It is not correct to state that the coefficient is a constant, over a range of duct sizes and air velocities. A very strong Reynolds N° influence may be anticipated, for this will determine the ratio of centre line (peak) velocity to mean $\left(\frac{q_v}{A}\right)$ velocity.

It has also been noted that conditions in most air systems are neither fully turbulent nor truly laminar but rather 'transitional'. Aside from frictional considerations, most easily determined by reference to the Moody chart (Fig. 1a), it will be appreciated that in conjunction with size, this will determine the thickness of the boundary layer. As a proportion of the duct diameter or equivalent, the variation may be enormous.

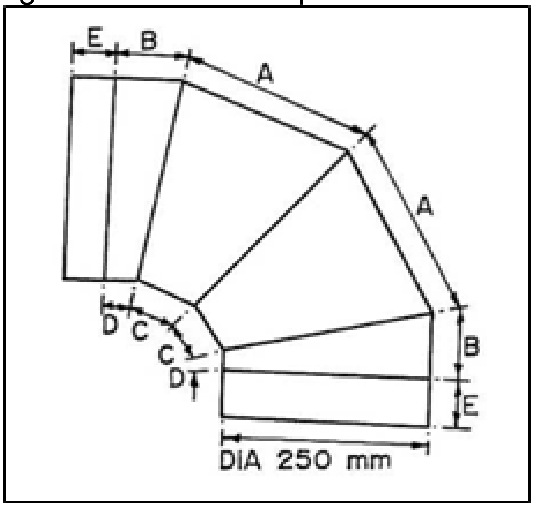
The above is merely an introduction to the vexed question of interaction effects when two bends or similar components are placed close to each other. If the bends are separated by a straight duct of say 30 diameters it may be expected that the loss of the two bends is the simple addition of the individual losses (plus of course the loss for the intervening straight duct). A little consideration of the velocity profiles will show why this is not the case in a typical working environment. Due to inertia forces (centrifugal and static pressure forces) acting on the peaky velocity profile at inlet to a bend, the core region is deflected outwards. Hence the velocity profile to the second bend is far from symmetrical and the loss coefficient for it is no longer valid.

Whilst the few text books and guides, which are aware of the problem, usually state that neglecting interaction effect may result in an over estimation of pressure losses, it should be expected that this is far from the case with tight bends $\left(\frac{r}{d} < 1\right)$ or mitred i.e., lobster backed bends. It should be especially noted that much of the data has been obtained from tests on water in smooth pipes where the flow is invariably fully turbulent. It is highly unlikely that it can be translated to the transitional or laminar flow in air duct systems without significant error.

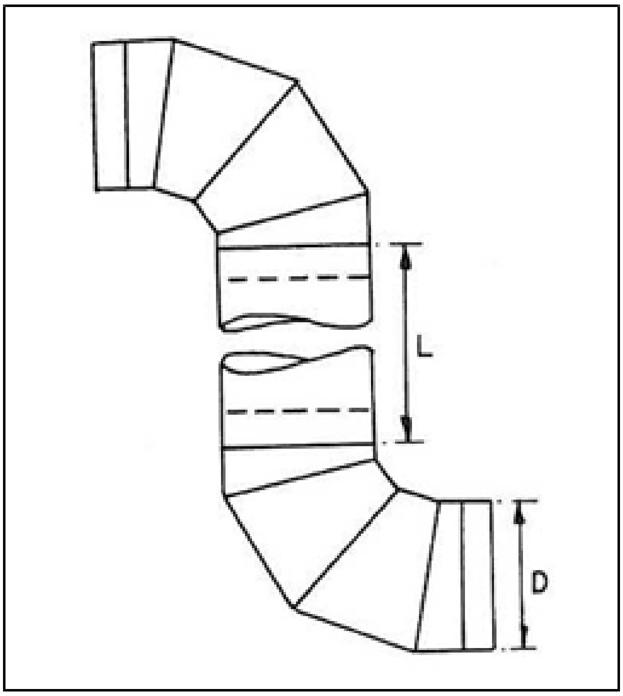
Finally it should be realised that the juxtaposition of two bends in a duct run can induce swirl downstream. This swirl may not decay for over 100 diameters of straight duct. Consequently the 'wetted' path is greatly increased in length and hence its pressure loss may be much higher than that calculated for the assumed straight line, fully developed swirl free flow.

5.2 Energy loss coefficients of two 90° elbows separated by a straight duct

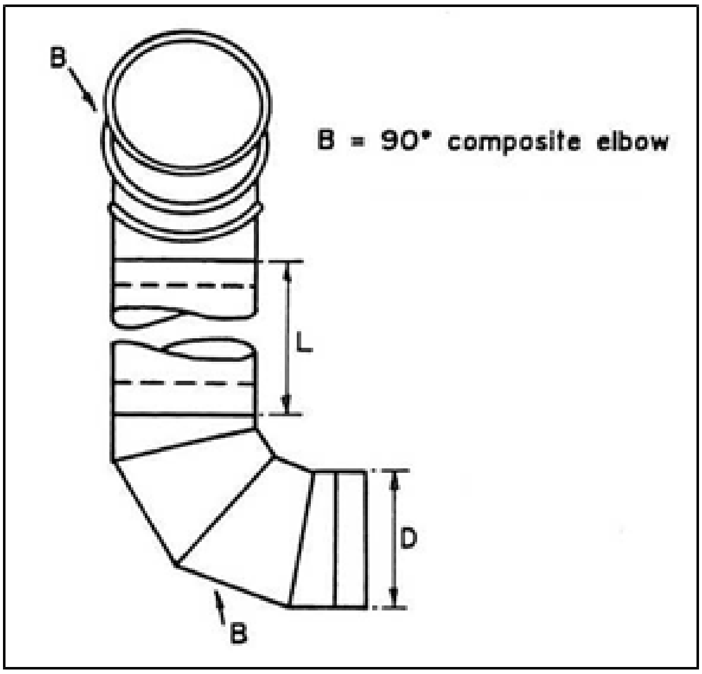
The results given below were obtained from a European Commission research Contract in June 1990 and have been published by Eurovent as: Measurement of mechanical energy loss coefficients - Part 1: Single and double composite elbows.



Single 90° composite elbow dia. 250 mm



Two 90° composite elbows in the same plane dia. 250 mm



Two 90° composite elbows out of plane dia. 250 mm

Fig. 6 Composition of lobster backed bends

5.3 Measured data

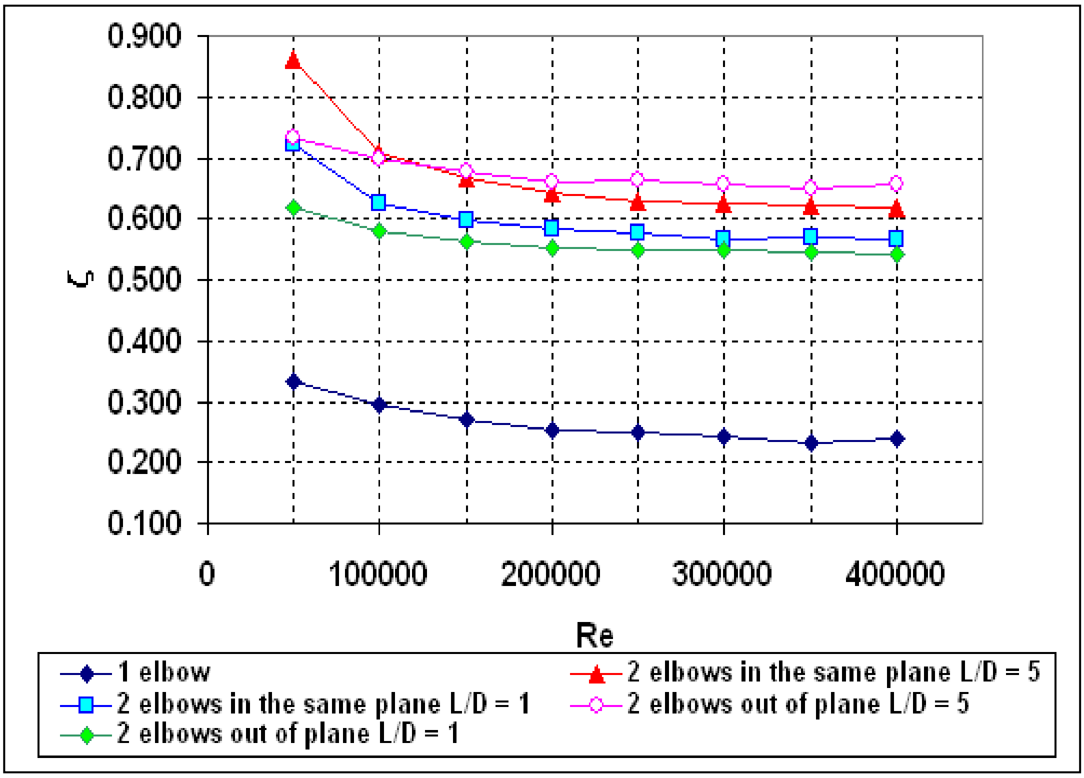


Fig. 7 Energy loss coefficient of 90° elbows (lobster backed bends)

5.4 Comparison between the measured and predicted data

The prediction of the loss coefficient is made with the following formula:

$$\varsigma_{\text{two elbows}} = 2.\varsigma_{\text{one elbow}} + \varsigma_{\text{straight duct}}$$

with:

$\varsigma_{\text{one elbow}}$: loss coefficient measured of a single elbow

$$\varsigma_{\text{straight duct}} = \left(0.005 + 0.42(\mathrm{Re}_D)^{-0.3}\right)L/D$$

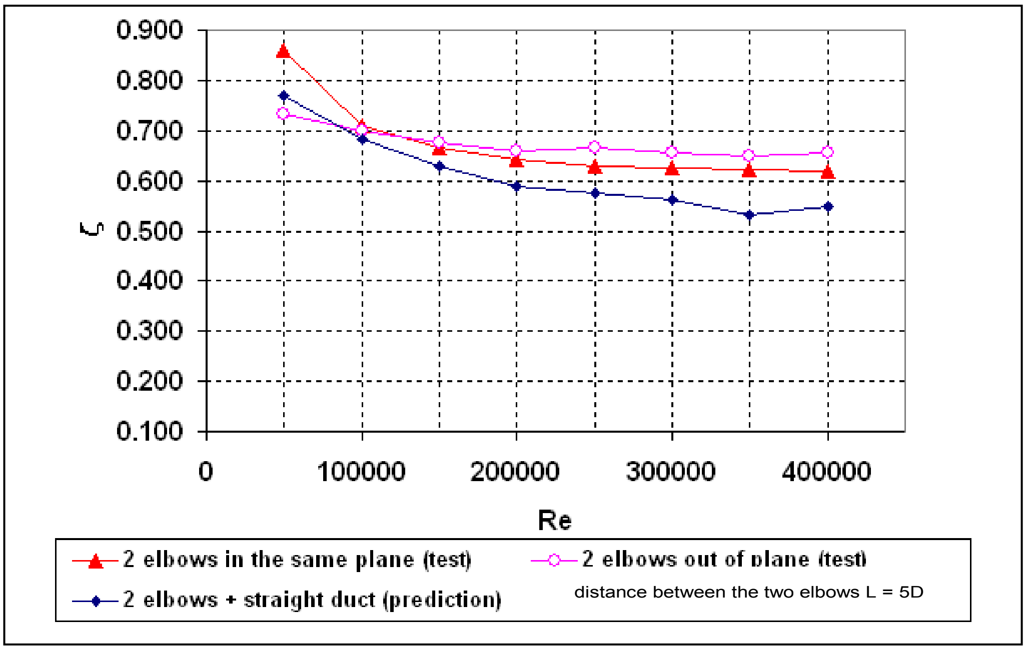


Fig. 8 Energy loss coefficient of 2 elbows separated by a duct L/D = 5

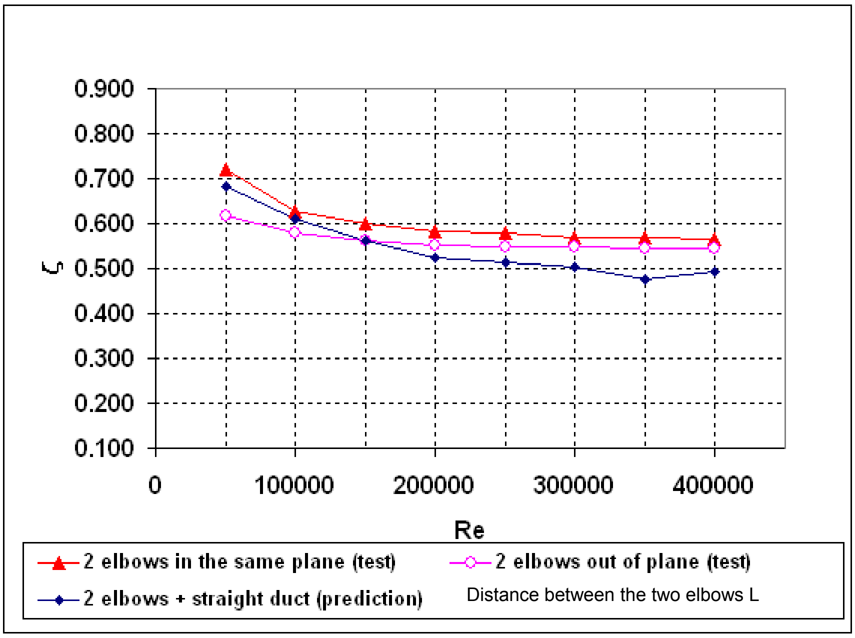


Fig. 9 Energy loss coefficient of 2 elbows separated by a duct L/D = 1

5.5 Conclusions

The energy loss coefficient of two elbows separated by a straight duct is not equal to the sum of the loss coefficients of the individual components. It may be slightly higher or lower and depends on the duct configuration, i.e. whether the ducts are in the same plane or not, the radius of the elbow, the length of straight between the elbow etc., etc.

Chapter 6: Upstream and Downstream Flow Conditions

Many of the problems with insufficient flowrate may be identified as those resulting from failure to appreciate that a 'system effect factor' may be present.

The most important concept, which has to be realised, is that fan performance differs according to how the fan is ducted. ISO recognises four different Installation categories :

A. where the fan is not ducted on its inlet and outlet, but where a pressure difference is maintained across a partition
B. where the fan has an open inlet but is ducted on its outlet
C. where the fan has an open outlet but is ducted on its inlet
D. where the fan is ducted on both its inlet and outlet

These categories are identified diagrammatically in Fig. 10 below

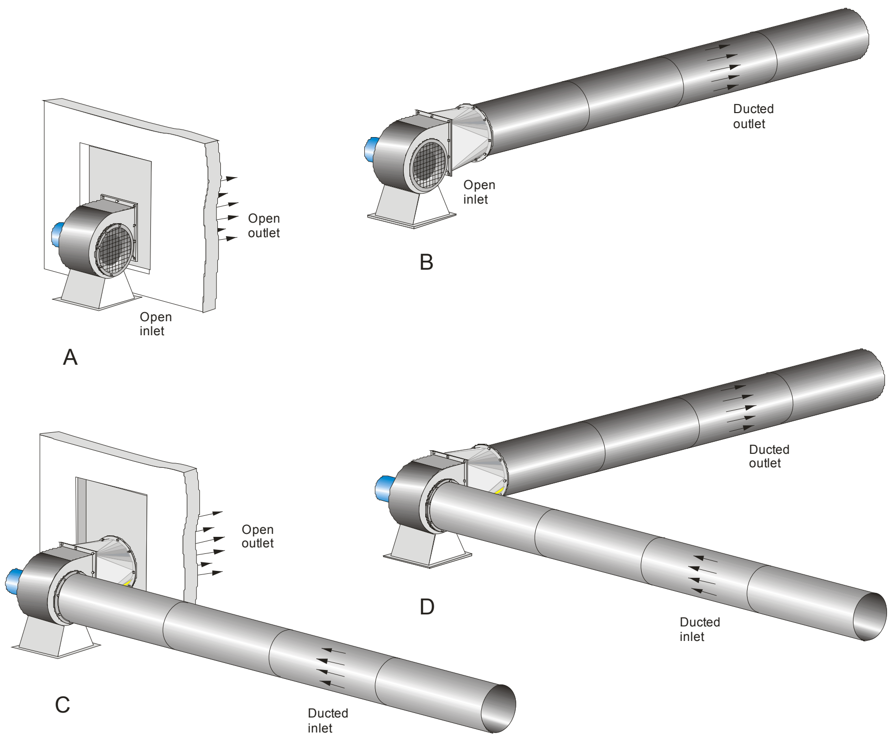


Fig. 10 Fan installation categories

For most ducted fans, catalogue performance is shown to Category D where the airflow is closely controlled and often gives the 'best' performance. The differences between the different categories are very much a function of the fan type and even its design. Typical performance curves are shown in Figs. 11 to 13 below. However the buyer should seek the appropriate information from the manufacturer by indicating the installation category in his enquiry.

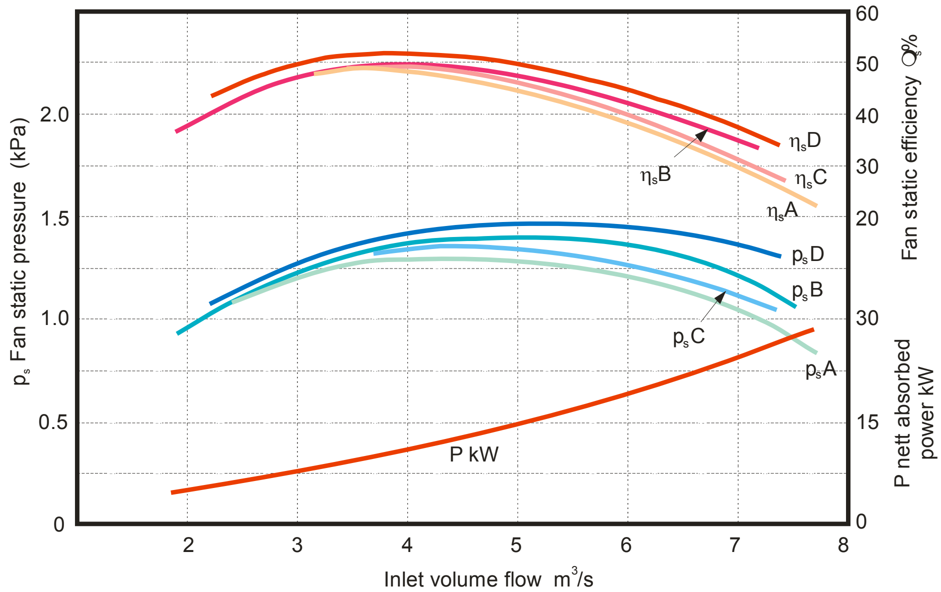


Fig. 11 Typical performance curves for a forward curved centrifugal fan to different installation categories

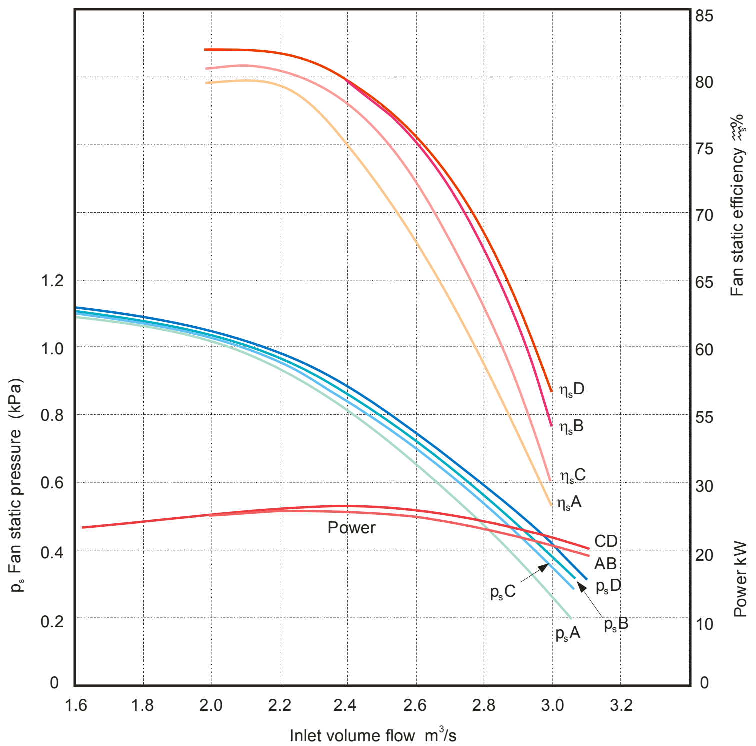


Fig. 12 Typical performance curves for a backward inclined centrifugal fan to different installation categories

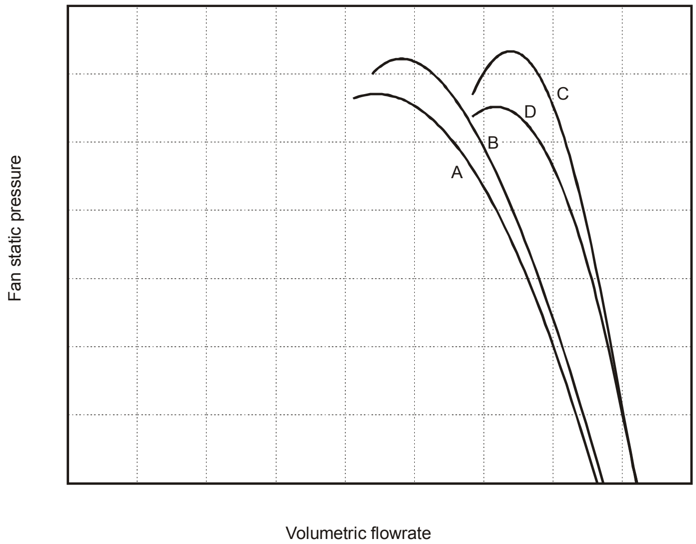


Fig. 13 Typical performance curves for a tube axial fan to different installation categories

It has been known by fan manufacturers for many years that ducting adjacent to the fan can have a considerable effect on the flowrate. The performance of the fan may be reduced and the pressure losses in this ductwork may be greater than normal. Whilst publications such as TM34 – Fan Application Guide (CIBSE/FMA 2007) have given advice on the design of fan connections indicating what were good, bad or indifferent, it has to be recognised that space is not always limitless. Consequently the ideal of, say, 5 diameters of straight duct, directly connected to the fan inlet and/or outlet is frequently not achieved.

Full catalogue performance of a fan is only attained when the conditions under which it has been tested are replicated. This requires that the flow at the inlet is fully developed, free from swirl and with a symmetrical velocity profile. On the fan outlet, there must be sufficient straight to enable the asymmetrical profile to diffuse efficiently into a fully developed symmetrical velocity profile further downstream. The cross sectional area of the ducts on the inlet and outlet of the fan must be nearly the same as the fan inlet and/or outlet area.

It has to be said that there is a lack of quantifiable data on such ‘system effect factors’. The most comprehensive and usable information is contained in ‘201 - Fans and Systems’ published by AMCA, although the basis for the data is sometimes outdated. The calculation of an additional, unmeasurable pressure loss, proportional to the velocity pressure, gives an acceptable solution in the working range of the fan. However it is far from accurate at static non delivery (closed) or at free inlet and outlet (fully open) conditions. The more limited information given in the ‘Fan and Ductwork Installation Guide’ (HEVAC/FMA) is then nearer the truth. ‘Fans and Ventilation’ by W.T.W. Cory (Elsevier 2005) devotes its Chapter 5 to a more detailed discussion of the whole topic.

It has been suggested by some that fan curves should allow for less than perfect connections. This is not possible – how bad would you like the connection to be?

Chapter 7: Swirling Flow and the Flow Profile

From all that has been said so far, it will be appreciated that calculated pressure losses in both straight ducting and fittings are only likely to be correct under certain well defined conditions. We are accustomed to reading that the flow must be fully developed, symmetrical and free from swirl. That these conditions are rarely achieved is frequently ignored.

A particular example is swirling flow. Tube axial fans (i.e. those without guide vanes) inevitably have swirling flow at their outlets. This swirl will vary across the fan characteristic. If therefore discharging into a long straight duct on its outlet, the resultant wetted path will be much longer than that for straight line flow. Thus the friction loss may be much greater dependent on the resultant distorted velocity profile and whether the impeller blades are designed for a forced or free vortex.

Losses in bends may be greater than normal whilst outlet losses may be smaller.

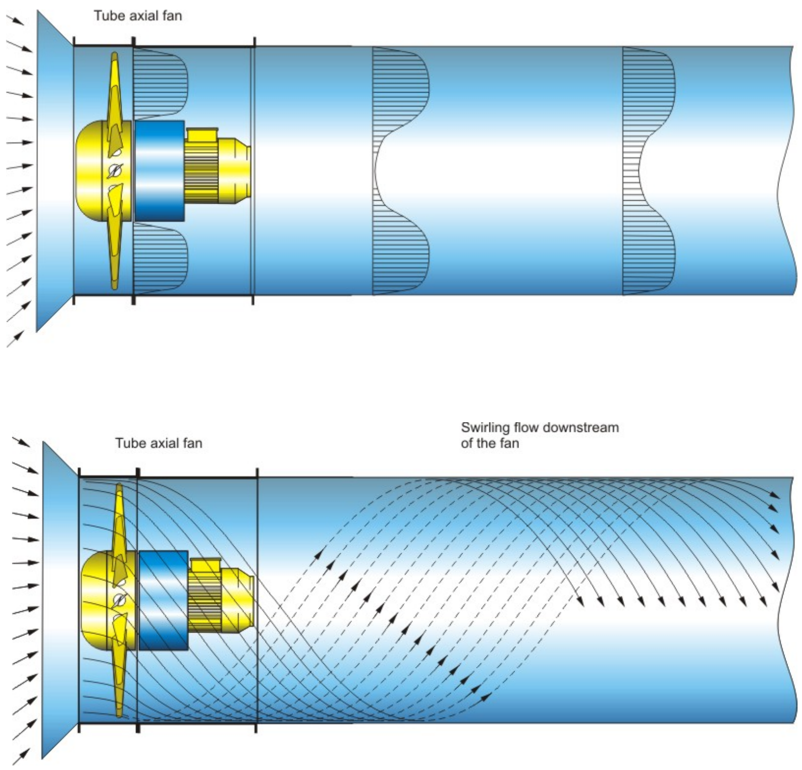


Fig. 14 Tube Axial Fan showing distorted velocity profile and Swirling flow downstream

It is therefore desirable that where axial fans are used on blowing systems, they should be equipped with guide vanes i.e., they should be Vane Axial fans. At their best efficiency point this will ensure virtually straight line flow (if designed to recover all the swirl energy) although there may be some residual swirl at operating points outside the design working range.

A more complex situation may exist at the outlet of Forward curved Centrifugal fans. Again if operating away from their best efficiency point dual contra-rotating vortices may be produced which proceed as contra-swirl along the duct.

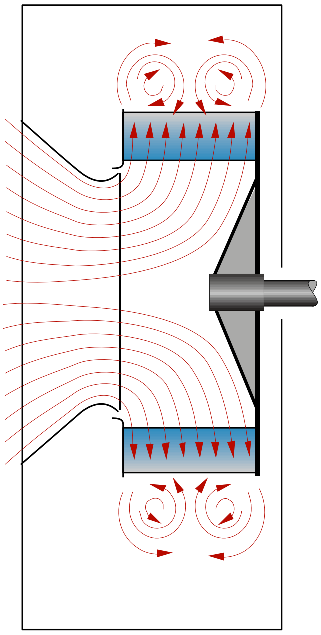


Fig. 15 Forward curved centrifugal fan showing contra swirl at the outlet from the impeller

It will be appreciated that in all the above cases, the increase in pressure loss above that normally calculated for straight line flow will be dependent on the angle of swirl and the 'peakiness' of the velocity profile. Flow distribution is most important. The situation for bends, junctions, cooling or heating coils, filters etc., etc., is even more complex. No information presently exists for calculating the real losses which may result.

It is important to know that swirl in a straight duct can continue for upwards of 100 duct diameters and may only decline very slowly.

Swirl is not only due to that existing at the outlet of particular types of fan. It can be induced by particular types of bend in series according to the angle of their juxta-position.

At the present time no resistance curves exist for systems with swirl – the variation in the degree of swirl, the type of fitting, the cross-section etc., make the variables too great in number to justify the necessary research programme.

Perhaps the only exception to the above statement is for 'C' type installations i.e., for systems where ducting only exists on the inlet side of the fan which discharges to atmosphere, any residual swirl then being dissipated.

Finally, a warning – a reduction in the diameter of a duct system already containing swirl will **increase** the swirl velocity and the associated losses.

Chapter 8: Differences between planned and built installations

Good quality designs may still be compromised at the time of their physical construction.

Most of this Eurovent document is intended to emphasize the importance of a correct prediction of the pressure loss, along a ventilation system, to prevent incorrect, inefficient or even unreliable operation of the system, and particularly of the fan.

Even so, a large number of the problems, related with incorrect predictions of the system pressure loss, do not emerge at the design-table stage, because of incorrect calculations, but later on, when the system is actually built up.

A number of last-minute changes may be introduced in the system, with the optimistic belief that they may not be important enough to have such a dramatic influence on the system. Some of these changes are introduced knowingly; sometimes they are just the result of lax installation procedures. Sometimes these changes are even applied with the true belief of improving the system, or its cost.

The wise system-designer should never relax in his attention to these details, and should always look carefully at the effect of these changes on the actual performance of the system.

The price for losing control of the actual installation process can be, in the worst case, a costly and painful process to re-gain not even the originally specified performance, but just an acceptable level of efficiency and reliability.

There is a wide range of reasons, leading to system changes, which can increase, by a larger or smaller amount, the predicted pressure loss.

Last-minute, and hastily engineered changes, may be needed to overcome routing problems not properly dealt with in advance, and may easily prove dangerous.

A clear example is the case of a ventilation duct, running along the ceiling of an industrial building, which, according to the original design, should have been running at some distance from the ceiling, just below the four pre-compressed concrete beams of the roof, extending below the ceiling.

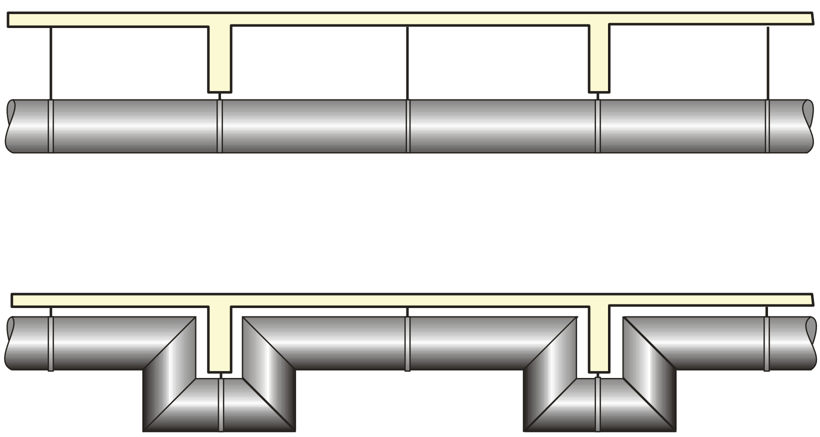


Fig. 16 Ventilation duct running along the ceiling of an industrial building

To increase the useful height inside of the building, the main duct was then raised against the ceiling in the middle parts of the building, introducing two neat inverse omega-shaped bends, to turn around each of the two beams running across the centre-part of the ceiling.

An attempt was made to predict the additional pressure loss of each of the two groups of four 90° bends, used to turn around each beam, and the fan was re-selected at the last minute, to compensate. Unfortunately, this is a classical case where close-coupled components do negatively influence each other, and the resulting pressure loss was considerably higher than four times that of a single bend.

Unsurprisingly, the real pressure loss of the entire distribution duct was considerably higher than predicted, preventing the system from achieving its originally specified duty. To satisfy the design requirement for volume flow, apart from replacing the simple bends with a design using flow-turning vanes, the fan had to be replaced with a different design, accepting increased power consumption, and accepting a higher noise level, from the up-rated fan, which was far from satisfactory.

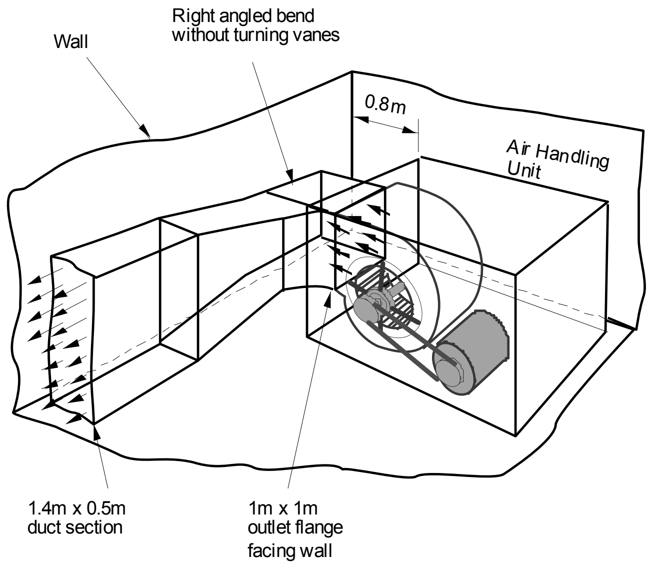
Residential and office buildings frequently give their own special problems in the design of the ventilation systems, because space for technical systems is normally at a premium. This leads frequently to less-than-ideal design of duct bends and transition pieces.

Odd-shaped transitions and bends are difficult to correctly account for, when making a pressure loss prediction, and may become awkward to predict, if made even sharper in their shape when the space constraints get worse during construction.

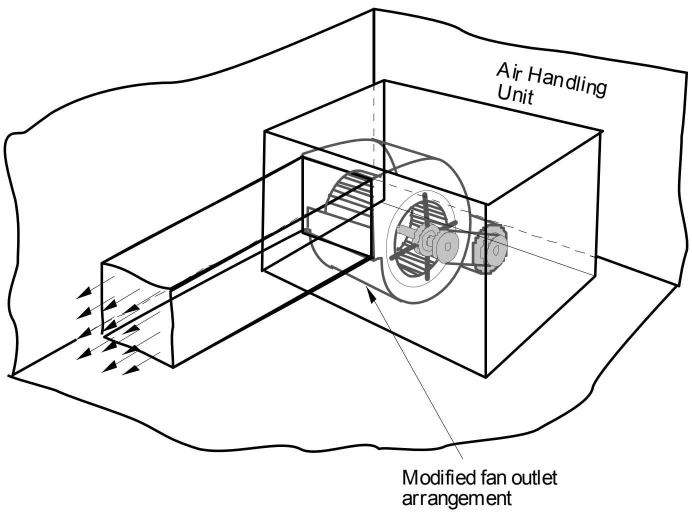
The transition pieces immediately downstream of an air handling unit are particularly critical, as they are frequently in close proximity with the fan.

The space reserved for installation of an air handling unit is normally kept as tight as possible, and often requires a 90° bend, immediately downstream of the A.H.U., typically to avoid a wall. This kind of installation is quite frequently shown to be too tight, when the final design of the AHU is completed, and ends up being just a bit longer then the first estimate.

In a specific case, the outlet of an AHU, ending with the fan section, was an 1000x1000 mm flange. A concrete wall was located right in front of the outlet, at a distance which, after the last changes to the A.H.U. design, was reduced to some 800 mm. Within this tight space, the airflow should be turned 90°, to run parallel to the wall. The selected design was a combined 90° elbow and contraction piece, without turning vanes, with a 200 mm inner radius, ending into a 1400 x 500 mm duct.



a)



b)

Fig. 17 Outlet of an air handling unit showing - a) Original duct - b) with modified design

The result of such an odd transition piece, located immediately downstream of an 800 mm double inlet forward curved fan, was an unpredictably high pressure loss, and a considerable disturbance to the working condition of the fan itself, with half the impeller running in a permanent stall condition. This led to repeated structural failures of the half impeller close to the inside of the bend, because of metal fatigue.

The actual volume flow of the system, because of this and other problems in the ducting system, was somewhere close to 50% of the original design duty.

The problem was solved only when the A.H.U. was re-built with a different final fan section, having the fan facing sideways, and discharging parallel to the wall, directly into the main duct of the ventilation systems, removing entirely the need for a bend. The entire process to identify and correct the problems required more than one year and a half, before the performance of the system could be considered fully satisfactory.

Ducting problems, arising at installation time, can also plague light-duty systems, using small, low pressure fans.

A common case of unpredicted pressure loss increase is linked with the use of round section, flexible ducts, made of fabric supported by a metal helical spring. These simple and inexpensive ducts are frequently used for the short connections linking a ducted fan-coil unit with the outlet louvers inside hotel rooms, but can be found also in other designs, especially when the connection is without branches and just few metres long. These ducts are frequently just laid down above the ceiling or on the counter-ceiling.

The problem arises when these ducts are sharply bent, e.g. when turning around corners or rising upwards and then turning downwards into the connection of the outlet louvre. If not properly laid down and supported, the flexible ducts can easily create folds in the fabric, considerably worsening the airflow in the bends, and increasing the predicted pressure loss.

The amount of the additional pressure loss created on such a short duct may seem negligible, but we should bear in mind that, in such a small “mini-system”, also the available pressure from the fan is frequently small, and the amount available to compensate the pressure loss outside the fan-coil unit itself may be really small as well. The relative effect of an incorrectly laid duct may thus prove important.

Some other types of changes in ventilation systems, which may prove dangerous for the reliability of the pressure loss predictions, are those introduced at a later stage of installation for “aesthetic” reasons, like a change in design of outlet louvre or nozzles. The selection of a physically different outlet design, even when introduced late, at installation time, is normally well accounted for. Still there are other, apparently innocent changes, which may prove trickier.

The last-minute decision to apply a special paint coat, to the outlet louvers of the fan coils installed in the ceiling of an exhibition hall appeared a negligible change; nevertheless, the application of a 0.6 mm thick, special, metalized paint coating, on the blades of louvers having very thin, 3 mm wide slots, increased velocity pressure for a given volume flow rate by a factor close to 3, increasing the pressure loss of the outlets roughly by the same factor. This was enough to bring the small fans inside the fan coils into a stall condition. The effect on performance was not critical, but the increase in low-frequency noise was strong enough. With 36 identical fans installed in the ceiling, their total effect was definitely annoying, and had to be corrected, requiring once again a long-lasting and expensive re-work.

This proves that even minor details should be carefully checked to avoid awkward surprises.

Chapter 9: Fan Noise Installation Effects

9.1 Introduction

When a fan is installed in a system or appliance its sound power level may be quite different from the one measured when it is mounted on a standardized test rig while operating at the same flow point. In the "ideal" case of the standardized installation the fan is often quieter than in situ and this difference in levels may be explained by an installation effect (or system effect) which depends on the fan and system design. This effect has to be taken into account to avoid unexpectedly high sound level on the actual installation. The difference in sound power levels of two fans tested under laboratory conditions may vanish or even there may be a hierarchy inversion in field tests, because of this system effect.

A review of the factors that can lead to fan noise installation effects is presented in [1] to [3]. Two main factors contribute to this mechanism: the acoustic loading of the system and the inflow conditions. The sound power radiated by the fan inlet and outlet depend on the acoustic impedances of both the fan and the system. The flow conditions at the fan inlet play a significant role since an inhomogeneous mean flow pattern in the fan inlet section may considerably increase the tones at harmonics of the blade passage frequency (BPF), while an increase of the turbulent velocity raises the broadband noise level.

9.2 Examples of fan noise system effect

9.2.1 Fan coupled to a heat exchanger

Several examples of system effects due to the presence of a heat exchanger in front of an axial or a centrifugal fan are shown in Chapter 10. These examples show that the distance between the exchanger and the fan as well as the location of the exchanger upstream or downstream of the impeller have a strong influence on the fan sound power spectrum.

9.2.2 Abrupt bend at the inlet of a tubeaxial fan

This example taken from [3] shows the influence of a 90° elbow at the inlet of a tubeaxial fan on its inlet sound power spectrum. The distance between the elbow and the fan (coloured in white in Fig. 18a) is varied from 0 to 3D, where D is the duct diameter (D = 350 mm). Fig. 18 shows the measured sound power spectra with the elbow successively at 0D, 0.5D and 3D from the fan inlet and with a straight duct of 4D length (reference configuration), for the same operating point.

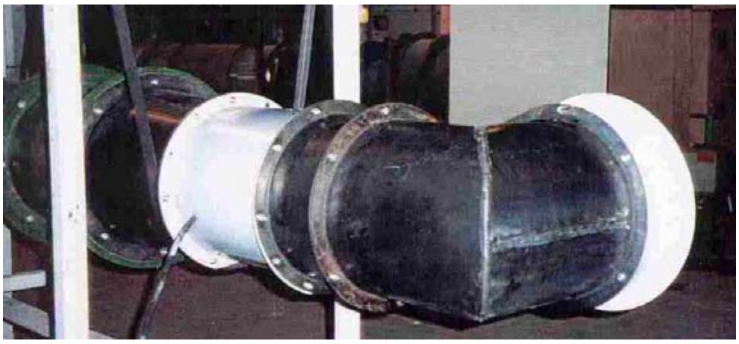


Fig. 18a Elbow at the fan inlet

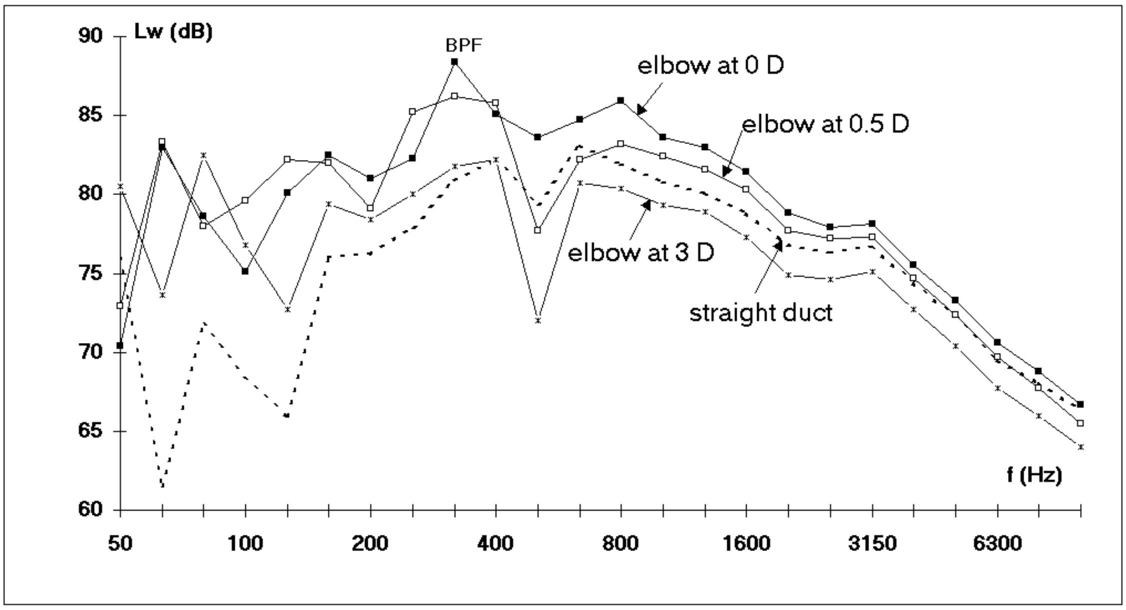


Fig. 18b Comparison of fan inlet sound power spectra for different inlet configurations

Fig. 18b shows large discrepancies between the spectra: at low frequency, the reference configuration is the quietest, while above 400 Hz the duct with the bend at 3D is the quietest. Above 400 Hz the noise level increases when the distance between the elbow and the duct decreases. The overall A-weighted inlet sound power levels for the different fan inlet configurations are shown in Table 2. The difference in levels between the quietest and the noisiest inlet setups is about 3 dBA.

Inlet Setup	LwA (dBA)
Straight duct	90.9
Elbow at 0D	92.7
Elbow at 0.5D	92.4
Elbow at 1D	91.3
Elbow at 3D	89.8

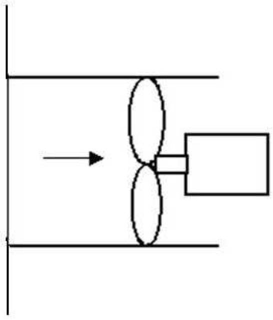
Table 2 A-weighted sound power levels with different inlet setups

Another example of noise installation effect induced by a box at the intake of an axial flow fan is presented in the next section. In this case a prediction of this effect is proposed and compared with experiment. This work is detailed in [4].

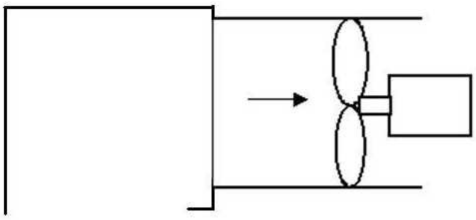
9.3 Prediction of fan noise system effect

9.3.1 Experimental set-up and test result

The tests have been performed on a tubeaxial fan of 400-mm diameter with 9 blades running at 2900 rpm. Two configurations of fan inlet are considered (See Fig. 19): the reference configuration (fan alone), which consists of a straight duct of 400 mm in diameter and 200 mm in length with a wooden baffle[1] and another configuration where the straight duct is preceded by a cubic inlet box of 405 mm edge with a lateral opening (fan + inlet box).

"Fan alone" configuration



"Fan + inlet box" configuration

Fig. 19 Fan inlet configurations

The fan outlet is fitted with a long straight duct ended by an anechoic termination and a reduced chamber with an orifice plate. This outlet ducting remains unchanged during the tests except the orifice plate diameter that is changed to get the same fan operating point in both configurations. The inlet sound power levels are measured in a reverberant room.

---

[1] A bevel edge has been cut in the wooden baffle at the junction with the duct, as guessed on the photo in Fig. 19

Fig. 20 compares the narrow band fan sound pressure spectra measured in the reverberant room with and without the inlet box at the same volume flow Qv = 1.45 m³/s. The inlet box strongly increases the noise level in the low frequency range up to 200 Hz and around 450 Hz, just above the blade passage frequency. Since the system effect mainly concerns the low frequency part of the spectrum the overall A-weighted inlet sound power level of the configuration "fan + box" is only 0.7 dB(A) higher than the level of the reference configuration.

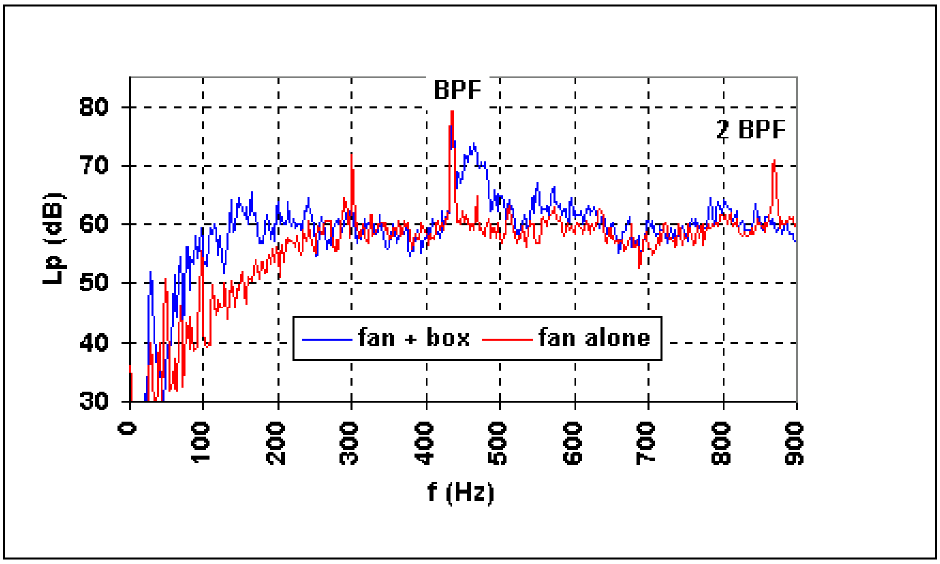


Fig. 20 Narrow band sound pressure spectra measured with and without the inlet box

9.3.2 Prediction of the acoustic system effect

As seen in the introduction the acoustic installation effect has two origins: acoustic loading of the system and deterioration of the inflow conditions.

The loading effect can be easily predicted in the low frequency range (i.e. below the first duct cut-off frequency, which is 530 Hz here) when the acoustic impedances of the fan and ductworks are known [4]. The acoustic impedances may be assessed from the measurement of the complex reflection coefficient of the elements, using a loudspeaker and two microphones. When the flow velocity is much lower than the sound speed, which is the case of many fans, the impedance may be measured with the fan not running.

The system effect due to inflow disturbances (non-homogeneity of the mean flow, increase of the turbulence level) is more difficult to predict with accuracy. The description of the model used here, based on work from Blake [5], is detailed in [4]. This approach requires knowing the RMS values and the integral scales of the axial component of the turbulent velocity in a duct cross-section close to the fan inlet. These data have been obtained from hot wire measurements in the straight duct cross-section at 50 mm upstream from the blade leading edge, for both configurations.

Fig. 21 presents three prediction curves: the blue curve shows the predicted system effect (difference between the sound pressure levels with and without the inlet box) due to the inflow disturbances, the red curve shows the predicted system effect associated with the acoustic impedance change between the two configurations and the green curve is the overall installation effect, which is the arithmetic sum in dB of the two other curves.

Beyond 200 Hz it can be noticed that the red curve has negative values in dB[2]. This result means that the impedance effect of the inlet box is beneficial, i.e. the box reduces the noise level of the fan in this frequency range.

The curve related to the turbulence injection effect is always above the 0 dB axis. This means that the increase of turbulence induced by the box leads to an increase in noise level in the whole frequency range between 50 and 550 Hz.

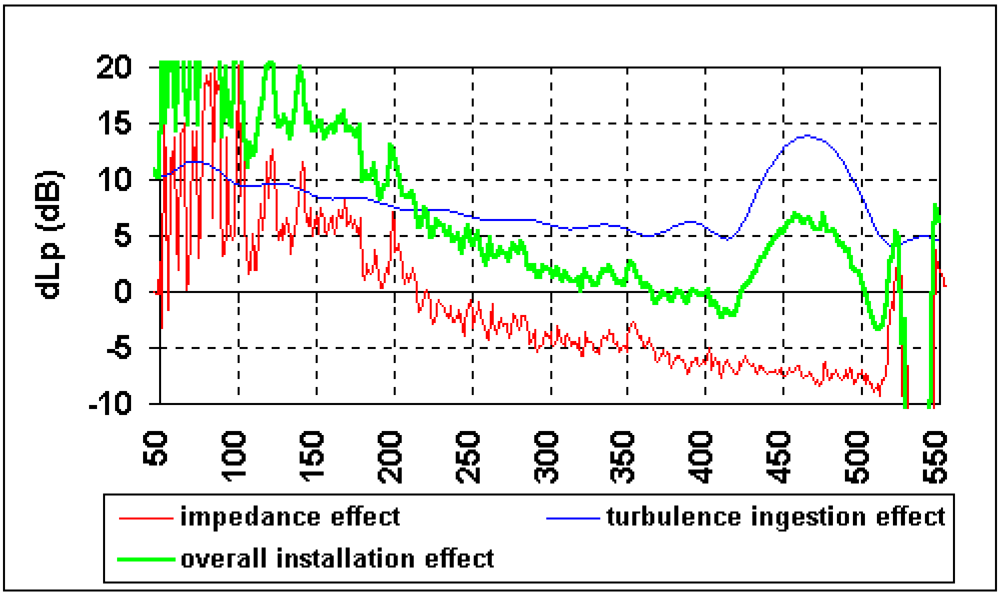


Fig. 21 Prediction of installation effect ($dLp = Lp_{with\ box} – Lp_{reference}$)

Fig. 22 compares the measured and predicted overall installation effect between 50 and 550 Hz. The prediction is quite good in the range from 100 Hz to 500 Hz. Below 100 Hz, where the acoustic loading effect is important, the experimental determination of the acoustic impedances is inaccurate and, therefore, the prediction is inaccurate too. Above 500 Hz the prediction of the acoustic loading effect fails as mentioned above.

2 The strong hole observed on the red curve between 500 and 550 Hz is caused by the failure of the prediction method above the duct cut-off frequency.

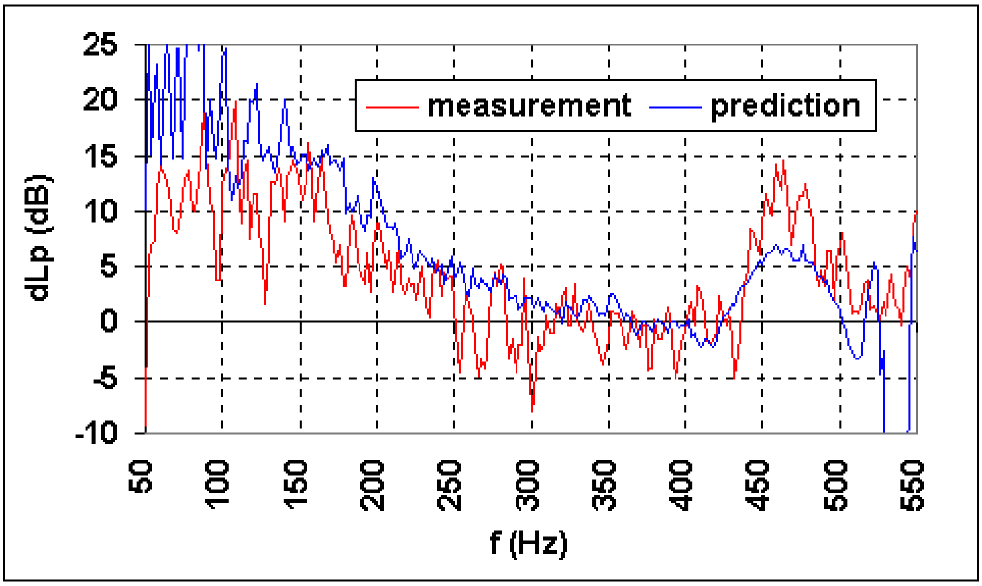


Fig. 22 Comparison of measured and predicted overall installation effect

9.4 Conclusions

The quantitative prediction of the fan noise installation effects is possible with reasonable accuracy as shown in the example presented here. The two mechanisms at the origin of the system effect, namely the acoustic loading effect and the deterioration of the inflow conditions, provide similar contributions to the overall system effect. The input data of the prediction models of these two mechanisms have been obtained from acoustic and hot wire testing, but it might be possible to get some of them from numerical simulations.

9.5 References

[1] A.N. BOLTON: Fan noise installation effects. Proceedings of the 1st Fan Noise Symposium, Senlis (1992)

[2] W. NEISE: Review of fan noise generation mechanisms and control methods. Proceedings of the 1st Fan Noise Symposium, Senlis (1992)

[3] A. GUEDEL: *Fan Acoustics – Noise Generation and Control Methods.* Edited by AMCA International, Inc. (2007)

[4] A. GUEDEL: Prediction of the noise installation effects induced by a bend at the inlet of an axial flow fan. Proceedings of the 2nd Fan Noise Symposium, Senlis (2003)

[5] W.K. BLAKE: *Mechanics of flow-induced sound and vibration.* Vol. II, Chapter 12. Edited by Academic Press, Inc (1986)

Chapter 10: Installation Effects of a Fan coupled to a Heat Exchanger

10.1 Introduction

When a fan is installed in front of an air-cooled heat exchanger, such as those used in air-conditioning or engine cooling systems, aerodynamic and/or acoustic fan system effects may occur due to the presence of the exchanger. These effects result in a reduction of the airflow and an increase of the noise level of the fan as compared with the performance of the fan tested alone at the same operating point. This deterioration of fan performance must be estimated and taken into account at the design stage to avoid trouble or disagreement. This paper shows some examples of system effects measured with axial flow fans and plug fans (centrifugal fans without volute) in different exchanger configurations.

10.2 Origin of the fan installation effects

10.2.1 Aerodynamic system effects

The experimental assessment of system effect on fan performance is made from the measurement of:

- the performance curve of the fan alone
- the performance curve of the fan within the system
- the pressure loss curve of the system.

Fig. 23 shows an example of results obtained on a cooling fan of agricultural machinery. The figure respectively shows the performance curve of the fan, measured on a standardised test rig (type A installation), the curve of the "fan + exchanger" actually measured (green curve in

Fig. 23 and the resistance curve of the exchanger[3]. The curve obtained from the difference between the fan curve and the exchanger resistance curve (pink curve in

Fig. 23 may be considered as the "fan + exchanger" curve without system effect. A strong difference is observed between the pink and the green curves, which means that the aerodynamic system effect is far from negligible in this case.

[3] The exchanger actually consists of five heat exchangers in series

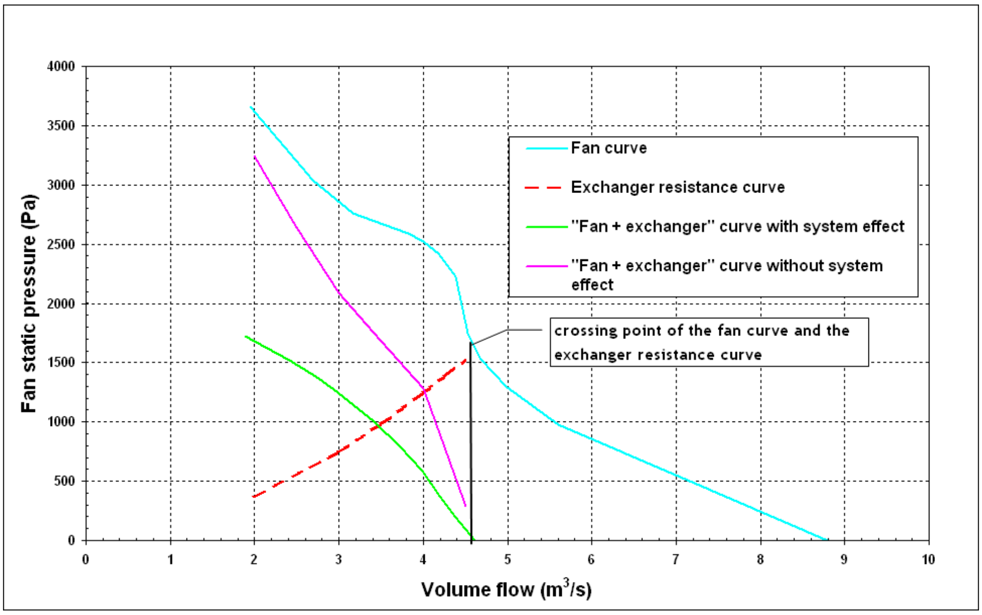


Fig. 23 Evaluation of installation effect on fan performance

According to [1] the three main causes of aerodynamic system effects are:

- non-uniform flow at the fan inlet
- swirl at the fan inlet
- flow distortion at the outlet.

In the present application the heat exchanger at the fan inlet may deteriorate the flow conditions in the impeller compared to those encountered in an "undisturbed" standardized test installation. When the exchanger is at the fan discharge the air velocity is far from uniform at the impeller exit, resulting in an underestimation of the exchanger pressure losses.

10.2.2 Acoustic system effects

Acoustic system effects have two main causes:

- deterioration of the inflow conditions, i.e. non-uniformity of the air velocity field and increase in turbulence of the airflow
- acoustic loading effect associated with the reflection of the sound waves radiated by the fan into the duct system.

10.3 Axial fan in front of a finned tube heat exchanger

Reference [2] presents results of an experimental study aimed at quantifying aerodynamic and acoustic fan system effects on a mock-up simulating an air-cooled condenser. A scheme of the experimental set-up is shown in

Fig. 24. A rectangular finned tube heat exchanger of 1.38 m² surface area is put in a box in front of an axial fan, either at the fan inlet or at the fan discharge. Four fans of different geometry and diameters varying from 710 to 775 mm were tested. The distance between the impeller and the exchanger was also varied from 0.35 to 0.75 D, where D is the impeller diameter.

The test results highlight the following trend in this case:

- system effects on fan performance are small whatever the fan geometry and the distance between the fan and the exchanger. In this particular case, a flow increase of 3 to 5% is noticed when the heat exchanger is at the fan discharge, which is probably due to a dynamic pressure recovery induced by the exchanger
- a significant acoustic system effect is observed up to about 500 Hz when the exchanger is at the fan inlet. An increase of the fan sound power level is observed in this frequency range in the presence of the exchanger for all four fans (
- Fig. 25). At higher frequency this effect considerably decreases.
- conversely, when the exchanger is at the fan discharge the acoustic system effect is quite negligible, or even negative below 125 Hz (Fig. 26). The difference in the results between the two positions of the exchanger is mainly due to the flow disturbance in the impeller when the exchanger is at the fan inlet.

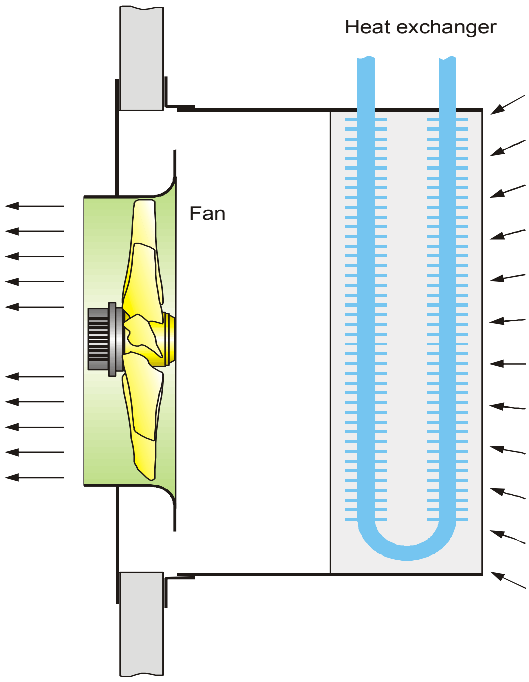


Fig. 24 Experimental set-up

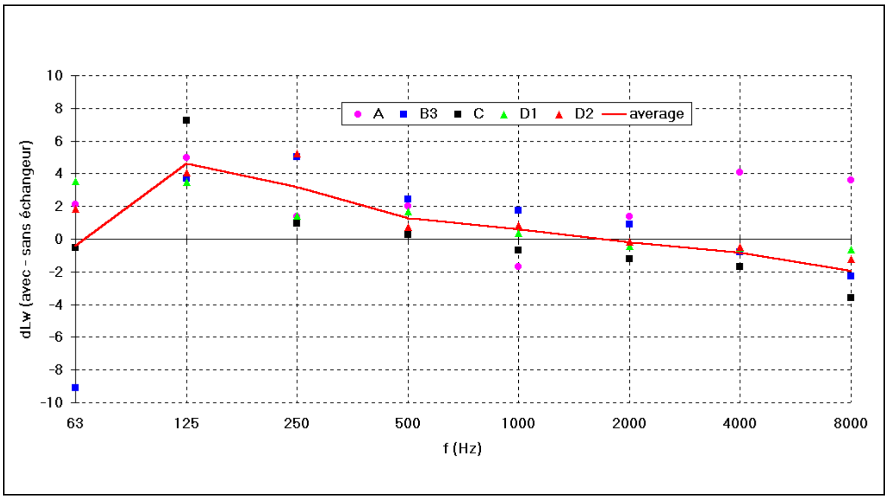


Fig. 25 Exchanger at the fan inlet $dL_W(dB) = L_{W \text{ with exchanger}} - L_{W \text{ without exchanger}}$

(L_W = sound power level radiated by the fan inlet + outlet)

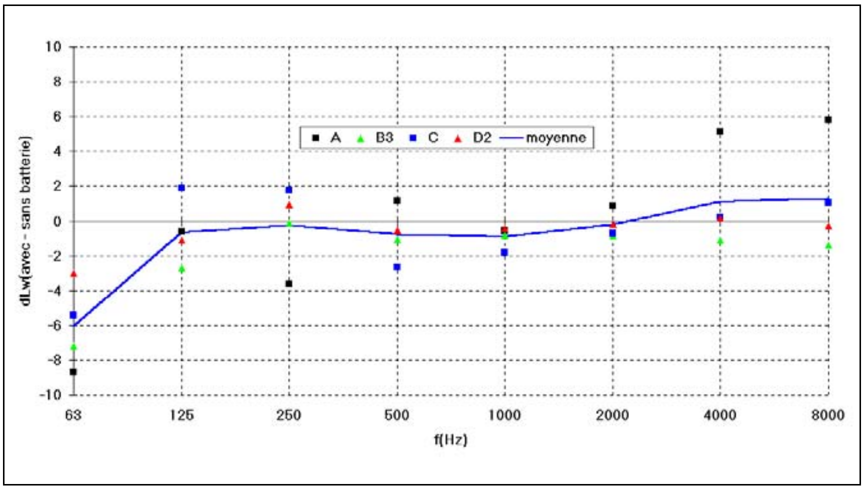


Fig. 26 Exchanger at the fan discharge $dL_W(dB) = L_{W \text{ with exchanger}} - L_{W \text{ without exchanger}}$

(L_W = sound power level radiated by the fan inlet + outlet)

10.4 Influence of the distance between a cooling fan and a radiator in agricultural machinery application

This example, taken from [3], shows the influence of the distance from a radiator to a fan on the airflow and A-weighted sound power level of the cooling unit. Three axial fans of diameter D between 600 and 700 mm and one centrifugal impeller of diameter 570 mm have been tested. The heat exchanger is always on the fan inlet side at different distances L to the fan. Fig. 27 shows that on the axial fans the flow slightly decreases with the reduction in fan-exchanger distance. On the centrifugal fan a significant flow reduction is noticed when the non-dimensional distance L/D is reduced from 0.2 to 0.1.

The noise level scaled at a constant arbitrary reference flow of 3 m³/s does not increase very much when L/D decreases from 0.4 to 0.2, but it increases quickly when this parameter is reduced further to 0.1 (Fig. 27).

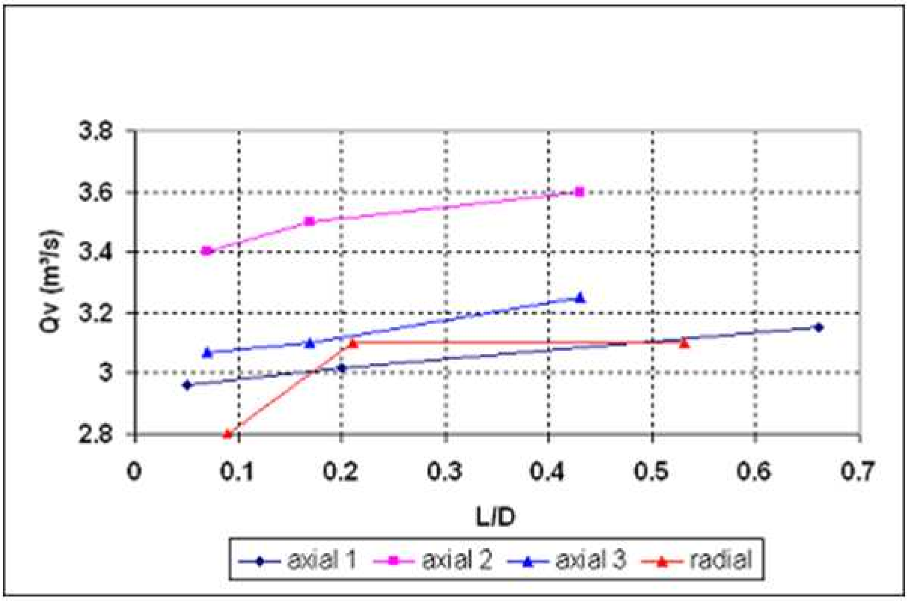


Fig. 27 Influence of the distance between the fan and the radiator on the airflow

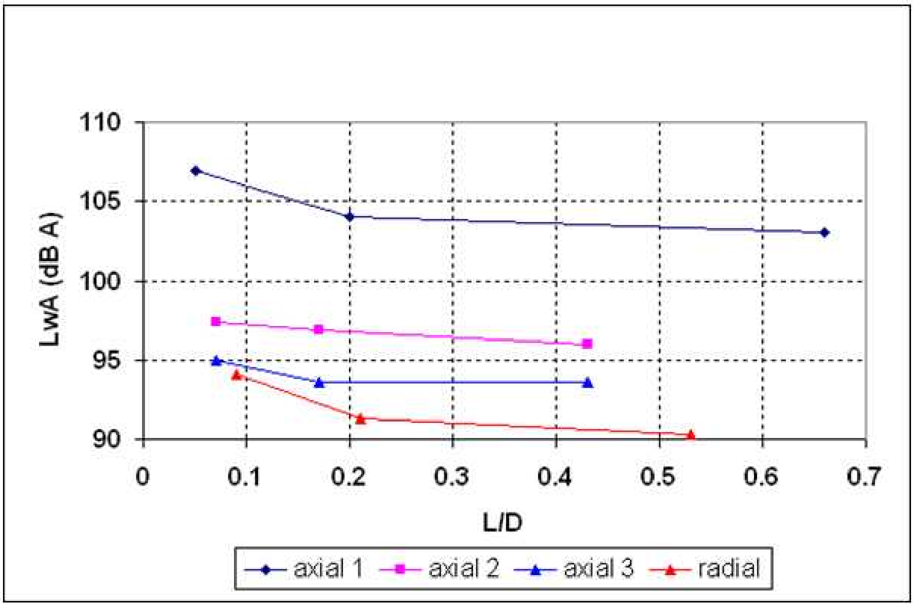


Fig. 28 Influence of the distance between the fan and the radiator on the A-weighted sound power level at constant flow Qv = 3 m³/s

10.5 Quantification of system effects of engine cooling fan on mobile machinery

In engine cooling unit of mobile machinery (digger for instance) the pressure losses of the heat exchangers installed in front of the fan are increasing due to the need for higher cooling capacity. In this application a centrifugal impeller without volute may be an interesting solution in replacement of the axial flow fan generally used, especially for reducing the noise level of the cooling unit. An important issue is the small space available in the engine hood, which limits the distance between the heat exchanger and the fan.

A series of tests have been performed at CETIAT on the cooling unit of a digger to quantify the importance of the aerodynamic and acoustic fan system effects when the distance from the upstream exchanger to the fan is varied. Fig. 29 shows a scheme of the unit including a heat exchanger, a backward curved centrifugal impeller and a transition section. The impeller diameter D is 508 mm and the non-dimensional distance L/D varies from 0.03 to 0.16.

Table shows the evolution of $\frac{\Delta Q_v}{Q_{v0}}$ and ΔL_{WA} with L/D[4]. $\Delta Q_v = Q_v - Q_{v0}$, where Q_v is the flow with the exchanger, Q_{v0} is the flow without system effect (i.e. flow measured on the fan alone at a static pressure equal to the system pressure drop). For each distance L/D, ΔL_{WA} is the difference in sound levels between the fan with and without the exchanger at the same flow.

A significant flow drop occurs when distance L decreases. The sound level is higher with than without the exchanger but the evolution of ΔL_{WA} with L/D does not show a very clear trend.

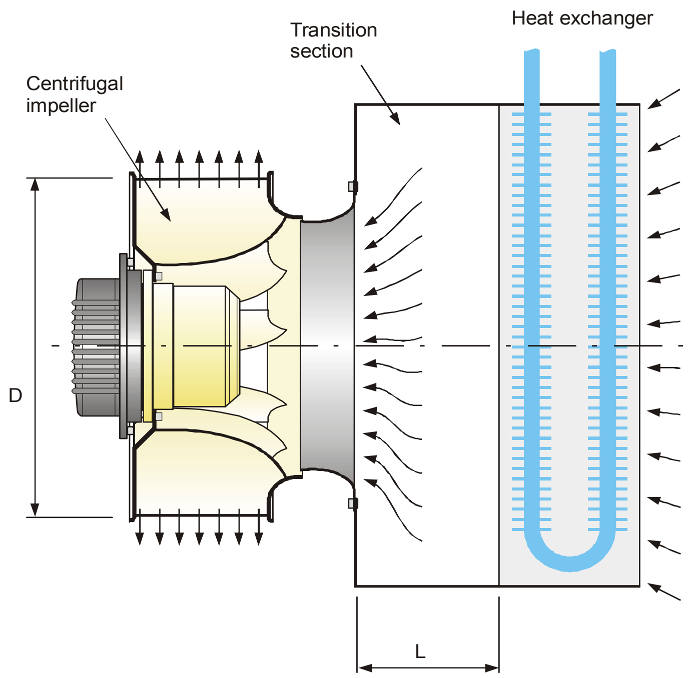


Fig. 29 Cooling unit set-up

L/D (%)	$\Delta Qv/Qv_0$ (%)	ΔL_{WA} (dBA)
3	-20	3
9	-11	5
16	-1	2

Table 3 Influence of distance L/D on the flow drop and noise level [4]

10.6 Conclusions

The aerodynamic and acoustic system effects of a fan installed in front of a heat exchanger are quite significant when the distance from the exchanger to the fan is lower than about 0.2 D, where D is the impeller diameter. This is true for axial and centrifugal fans. It is therefore recommended to keep a distance of at least 0.2D between these two elements. Furthermore, it seems preferable to install the exchanger at the fan discharge rather than at the fan inlet whenever it is possible.

10.7 References

[1] AMCA Publication 201-02 "Fans and Systems" (2007)
[2] A. GUEDEL: Installation effects of axial flow fans coupled to a heat exchanger. ImechE Seminar Publication 1997-4 (1997)
[3] Y. GOTH, M. LASSALAS: Acoustic and aerodynamic interaction between a fan and a heat exchanger. Proceedings of Fan Noise 2007 Symposium (2007)

[4] The test data in Table 3 have been obtained on two fans of same diameter and different geometry. Averaged results of the two fans are presented in the Table.

Summary and Conclusion

There is a widespread belief that pressure losses and noise generation in aerodynamic systems can be pre-calculated with a fair degree of accuracy. This paper shows that this is not the case and flow patterns are much too complicated to allow this.

It is shown that in most installations the flow is neither laminar nor fully turbulent but at an intermediate state. The resistance coefficients given in literature, which form the basis of all calculations, do not normally take different flow modes or patterns into account at all and therefore cannot be expected to be very accurate.

The flow directly downstream of a bell-mouth is quite uniform and has the same velocity in the entire cross section. The energy used to achieve this flow i.e., the inlet resistance is the lowest possible. But friction losses further downstream are high, as the fluid velocity at the wall is high.

On the other hand the flow after a long straight section is peaky with high velocity in the centre and low velocity at the walls. This means it has a higher kinetic energy but will have lower friction losses further downstream.

It is also shown that the pressure losses in singularities such as bends are highly dependent on the flow pattern entering it, i.e., the upstream configuration, so they cannot be considered independent entities.

Different flow patterns will also cause different noise levels.

This limitation in accuracy should not discourage designers from performing these calculations but they should be prepared to make allowance for the possible necessity to make corrective changes in the system, such as changing impeller pitch, speeds or other characteristics.

Annex A: System resistance – Interaction with combustion

A.1 Combustion

Defined in simple terms, combustion is the chemical combination of oxygen with combustible material such as carbon, hydrogen and, if unavoidable, sulphur. Oxygen is of course a constituent of the air around us. Under normal ambient conditions, air contains about 21% oxygen by weight. The remaining 79% however is almost entirely composed of nitrogen which to all intents and purposes is inert.

Before combustion actually takes place, a solid fuel must be heated to ignition temperature. The volatile gases in combination with the oxygen in the air supply then burn, and by increasing the temperature of the remaining material, ignite the fixed carbon. This is converted into carbon monoxide or carbon dioxide, according to the amount of oxygen present. Any non-combustible material remains as ash. It should be noted that pulverized coal generally burns firstly, by the formation of carbon monoxide (and other volatile distillates) and then further to carbon dioxide.

For liquid fuels the combustion process is simpler. They are soon converted into gaseous compounds, which burn very much as gases proper. Gaseous fuels burn immediately and do not have the severe problems of an ash residue. They may however produce significant quantities of moisture in the form of water vapour.

When fuels are burned, the whole of the heat produced cannot be used. Apart from furnace radiation losses, some of the heat is taken up by the products of combustion.

Perhaps most importantly there will be a considerable loss due to the amount of excess air used in an endeavour to obtain complete combustion. For this reason alone the use of mechanical draught is now almost universal. The theory of combustion is comparatively simple. It is much more difficult to apply in practice, however. Properly mixing air in the correct proportions with fuels and combustible gases to obtain complete combustion is not easy. Often the quantity of air delivered to the furnace is far in excess of that theoretically required. Although excess air means a loss of boiler efficiency, it is often necessary to ensure complete combustion, the amount depending on the quality, quantity and size of the fuel burnt. When fan draught is used, the air supply can be closely regulated and controlled.

A.2 Operating advantages

Whilst the numbers of boiler plant continuing to use natural draught alone is now very small, it is as well to remember the advantages that are obtained with the use of mechanical draught fans:

increased boiler output and reduced heat losses via the chimney lower grade and less costly fuel may be used exact adjustment of draught to boiler load requirements improved combustion obtainable which with proper firing will reduce smoke emissions permits the addition of heat recuperating equipment such as economizers and air pre-heaters to reduce exit gas temperatures and therefore heat losses.

A.3 Determining the correct fan duty

The carbon dioxide percentage in the flue gases at exit from the boiler is a measure of the excess air admitted for combustion. It is dependent on the average maximum theoretical CO_2 % of the particular fuel being burnt and the method of firing.

It may be assumed that a lower CO_2 % corresponds to "poor" combustion whilst a higher figure is 'good'. Medium percentages would be 8 to 10% for coal, whilst 5 to 8% would be considered "poor". The figures for oil are closer to their theoretical maximum, reflecting increased ease of obtaining good combustion with this fuel.

For more detailed information concerning particular boiler types and other fuels you should consult the boiler manufacturers.

Note: *With modern boiler types such as the condensing boiler, efficiencies exceeding 90% are possible.*

In industrial processes system resistance is obviously affected by combustion. The main reasons could come from the fact that gas is produced by the combustion itself, with a gas composition and a temperature which may vary, but also from the additional air which may be introduced to be diluted in order to regulate the temperature or for the combustion itself.

Other parameters can be even more significant: this is the case of Circulating Fluidized Bed boilers which are widely used for multi-fuel combustion of waste. When several fuels having varying heat values are burned simultaneously the boiler control system becomes very challenging as it is not possible to measure the energy content of the multi-fuel flow.

But the main characteristic of the system resistance of fluidized beds concerns the fluidization process itself as illustrated by the different phases on the following pictures when air velocity increases:

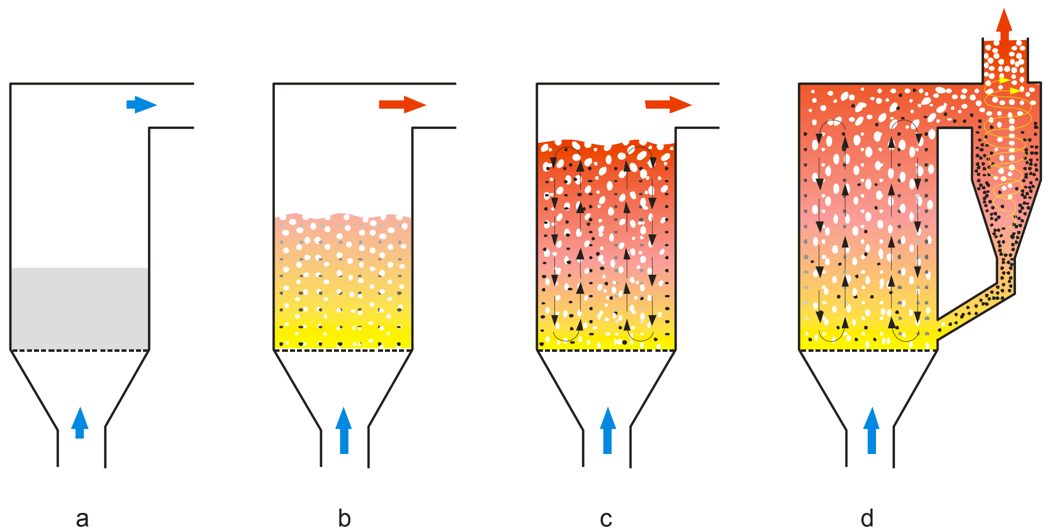


Fig. A1. Fluidization phases.
When the fluidizing air or gas velocity is increased, the gas-/solid-contacting mode moves from fixed bed (a) to bubbling bed (b).
If the fluidizing air velocity is further increased, the bed transforms into turbulent bed (c) and finally to circulating bed (d) (Adapted from Raiko et al. 1995).

When a bed of fine particles is subjected to an upward stream of air, the particles become suspended as the airflow reaches a certain velocity. This condition is referred to as the minimum fluidizing velocity and it varies according to the particle size and the depth of the bed. When the bed is fluidized it resembles a boiling liquid. Such a turbulent mass of solid particles is named a fluidized bed. Coal can be fed into the bed and as it burns it resembles molten lava. A typical pressure drop curve as a function of the air velocity is represented below:

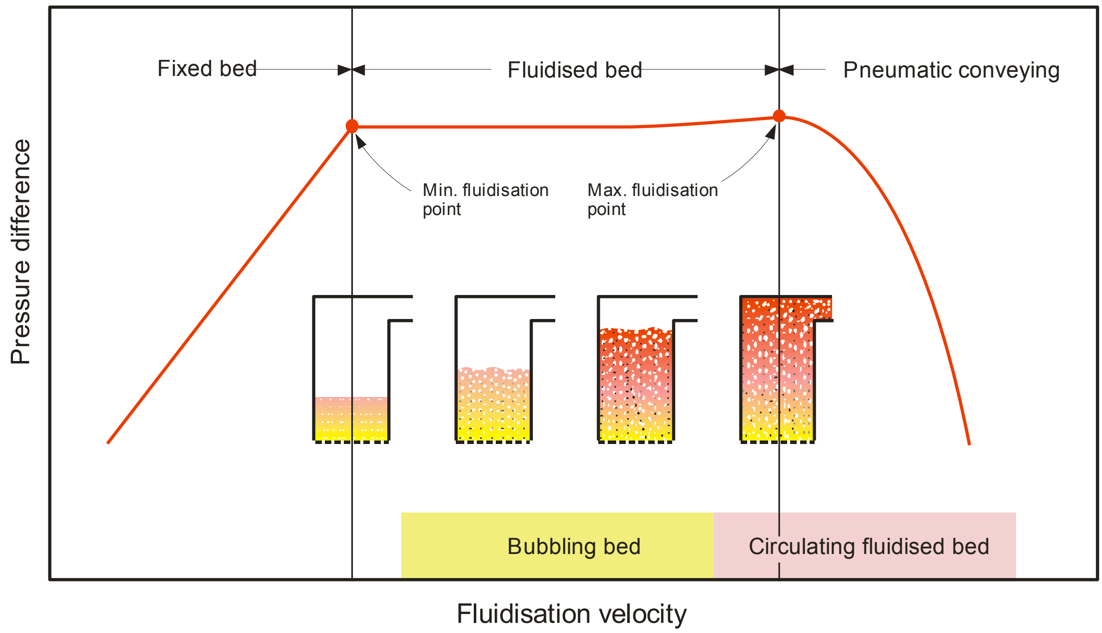


Fig. A2 Pressure loss in the flow through bulk material

The following illustration shows particle motion inside a fluidized bed. This recirculation affects directly the system resistance which cannot be calculated with gas consideration only.

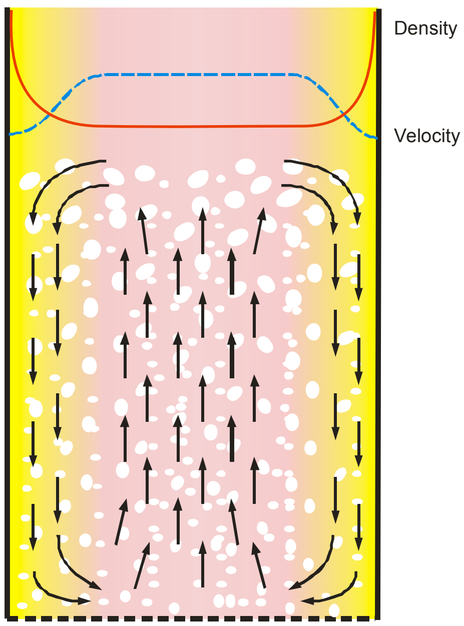


Fig. A3. Typical particle motion inside the circulating fluidized bed reactor.
The solid density increases near to the walls because the gas velocity is lower. Part of the particles combine together to form particle agglomerates known as clusters.
These clusters are falling downwards and form internal material circulation inside the reactor or combustion chamber. (Redrawn from Raiko et al. 1995.)